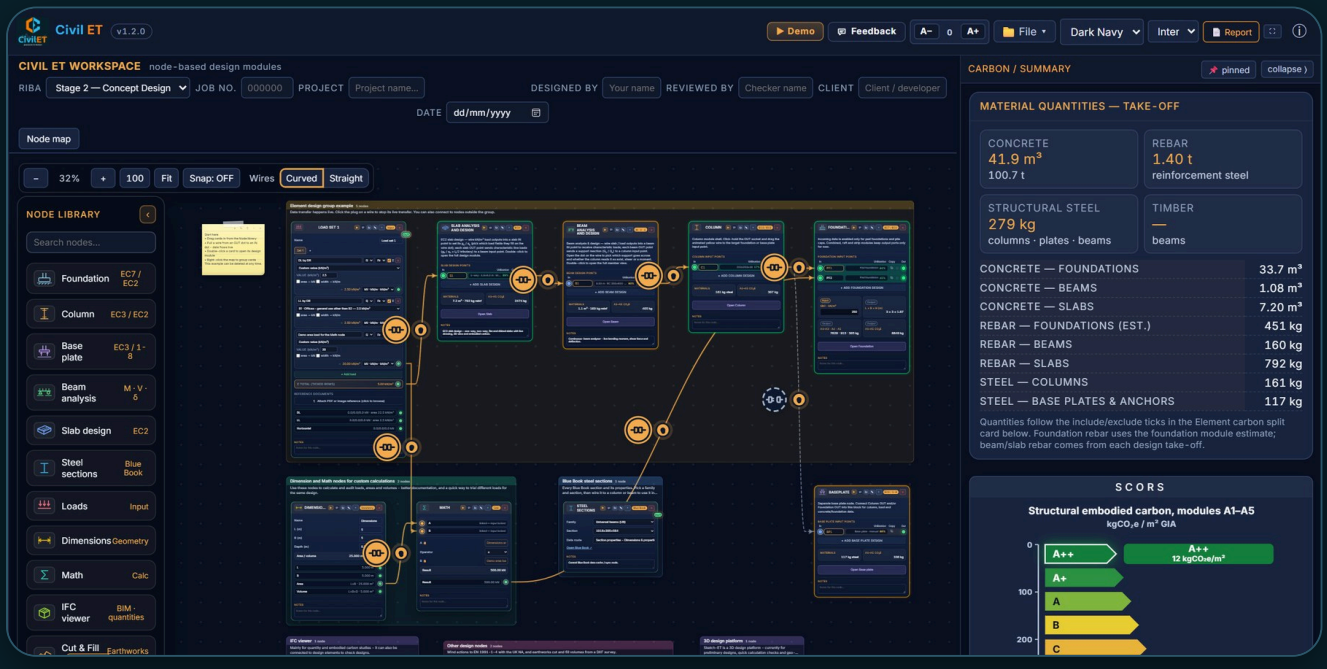
CivilET

precision in motion

Civil ET
User Guide

A guide to the node map, the design modules and the helper tools, with screenshots, step-by-step answers, troubleshooting, a glossary and an index.

Version 1.2.0 | Eurocodes + UK National Annexes | October 2026



Contents

Chapter 1 Introduction and Getting Started 4

1.1 What Civil ET is 4
1.2 What is new in version 1.2.0 4
1.3 System requirements 5
1.4 First launch 5
1.5 A tour of the screen 6
1.6 The 15 node types at a glance 8
1.7 Tips and known caveats 11

Chapter 2 The Workspace and Node Map 12

2.1 Overview 12
2.2 Adding nodes 12
2.3 Ports, wires and links 13
2.4 Selecting, moving, aligning and grouping 15
2.5 Deleting nodes - what else goes 16
2.6 Moving around the map 16
2.7 The right panel on the node map 17
2.8 Undo and redo 18
2.9 Saving, loading and exporting 18
2.10 The example workspace 19
2.11 Sticky notes 20
2.12 Live load sets 21
2.13 Project details and RIBA stages 21
2.14 Feedback, help and the tour 22
2.15 Performance and level of detail 22
2.16 Start-up crash guard and backups 23
2.17 Tips and limitations 23

Chapter 3 Foundation designer 24

3.1 Overview 24
3.2 Opening the designer 24
3.3 Foundation types and how they are modelled 25
3.4 Inputs - pad 26
3.5 Calculation - what is checked 29
3.6 Outputs 32
3.7 Multiple foundations and the Foundation node 33
3.8 Reports 33
3.9 Tips and known caveats 33
3.10 Screen reference - panel by panel 34

Chapter 4 Column design 47

4.1 Opening and adding a column 47
4.2 Load model (load take-down) 47
4.3 Steel columns (EN 1993-1-1) 49
4.4 Reinforced concrete columns (EN 1992-1-1) 50
4.5 Quantities and carbon 52
4.6 Results and reports 52
4.7 Alternative-section searches 53
4.8 Connections to other nodes 53
4.9 Tips, limitations and caveats 54
Screen reference - panel by panel 54

Chapter 5 Base plate and anchor design 64

5.1 Opening and adding a base plate 64
5.2 Choosing a calculation basis 64
5.3 Inputs 64
5.4 What the calculation does 66
5.5 Results 68
5.6 Report 68
5.7 Connections 69
5.8 Tips and caveats 69
5.9 What changed in v1.2.0 70
Screen reference - panel by panel 70

Chapter 6 Beam analysis and design 74

6.1 Opening and adding a beam 74
6.2 The structural model 75
6.3 Analysis modes 76
6.4 Load combinations (EN 1990) 77
6.5 Concrete beam design (EC2) 78
6.6 Steel beam design (EC3) 82
6.7 Timber beams 83
6.8 Quantities and embodied carbon 84
6.9 Connections to other nodes 85
6.10 Results, reports and the node card 85
6.11 Tips, limitations and known caveats 86
Screen reference - panel by panel 86

Chapter 7 Steel sections, Loads, Dimensions and Math nodes 96

7.1 Steel sections (Blue Book) 96
7.2 Loads node 99
7.3 Dimensions node 101
7.4 Math node 102
7.5 Which wires are allowed 103
7.6 Tips, limitations and known caveats 103

Chapter 8 Slab design 104

8.1 Opening and adding a slab 104
8.2 Inputs 104
8.3 Analysis 107
8.4 Design checks 109
8.5 Optimiser 111
8.6 Outputs and results panels 111
8.7 Reports 111
8.8 Quantities and embodied carbon 112
8.9 Connections to other nodes 112
8.10 Tips, limitations and known caveats 113
Screen reference - panel by panel 113

Chapter 9 Wind designer 119

9.1 What it does 119
9.2 Opening the node 119
9.3 Inputs 120
9.4 Describing the building 124
9.5 What the calculation does 124
9.6 Views 126
9.7 Outputs 127
9.8 Reports 128
9.9 Checking against a hand calculation 128
9.10 Connections to other nodes 128
9.11 Tips and limitations 128
Screen reference - panel by panel 129

Chapter 10 Sketch-ET 141
10.1 What it does 141
10.2 Opening it 141
10.3 The screen 141
10.4 Ribbon tabs 142
10.5 Setting up 142
10.6 Drawing and working 144
10.7 Templates 146
10.8 Calculation basis 146
10.9 Results 147
10.10 Site location and the Google 3D map 147
10.11 RIBA stages 148
10.12 Reports and exports 148
10.13 Connections to other nodes 148
10.14 Tips and limitations 149
Screen reference - panel by panel 149

Chapter 11 IFC Viewer and Embodied Carbon 156
11.1 Overview 156
11.2 Adding the node and loading a model 156
11.3 Categories and material buckets 157
11.4 The module screen 159
11.5 A1 to A3: product stage factors 160
11.6 Reclassification and reinforcement allowance 161
11.7 A4 and A5: transport and construction 162
11.8 Manual additions 163
11.9 Workspace carbon summary and SCORS rating 164
11.10 Outputs, ports and wiring 164
11.11 Comparing models 165
11.12 Designing from the model 165
11.13 Known limitations 165
11.14 Report 166
11.15 Tips 166
Screen reference - panel by panel 166

Chapter 12 Cut and Fill 171
12.1 Overview 171
12.2 Adding the node and opening the module 171
12.3 Importing the DXF 172
12.4 DXF layers 173
12.5 Point review 173
12.6 Ground model (TIN) 174
12.7 Surface display 174
12.8 Excavation shapes, footprint and formation level 175
12.9 Volume method 176
12.10 Outputs 177
12.11 Report 177
12.12 Connections to other nodes 177
12.13 Tips and limitations 177
Screen reference - panel by panel 178

Chapter 13 SI map 183
13.1 The SI map card 183
13.2 Layout of the module 184
13.3 Adding reports 184
13.4 The map 185
13.5 Reviewing a report 186
13.6 The summary panel 187
13.7 The heat layer and its check 187
13.8 Library and privacy 187
13.9 Tips and limitations 187

Chapter 14 Reports 187
14.1 Overview 188
14.2 Producing a report 188
14.3 House style 189
14.4 Gating - a report prints only what the design contains 190
14.5 What each module's report contains 191
14.6 The combined (node map) report 192
14.7 Tips and caveats 193

Chapter 15 Checking Your Results 194
15.1 Engineer's responsibility and independent checking 194

Chapter 16 Release Notes 195
16.1 Version 1.2.0 195
16.2 Earlier versions 197
16.3 Notes 197

Chapter 17 References and Data Sources 198
17.1 Design standards (Eurocodes and British Standards) 198
17.2 Anchors and fastenings 198
17.3 Steel design guidance 199
17.4 Concrete, geotechnical and loading guidance 199
17.5 Embodied carbon 199
17.6 Ordnance Survey and other data sources 199
17.7 Notes on use 200

Appendix A Quick answers: how do I...? 201
A.1 Workspace, node map, projects and reports 201
A.2 Foundations, columns and base plates 208
A.3 Beams, slabs, loads and steel sections 220
A.4 Wind, Sketch-ET, IFC and Cut & Fill 231
A.5 SI map 248

Appendix B Troubleshooting 250
B.1 Workspace, node map, projects and reports 250
B.2 Foundations, columns and base plates 252
B.3 Beams, slabs, loads and steel sections 254
B.4 Wind, Sketch-ET, IFC and Cut & Fill 256
B.5 SI map 259

Appendix C Glossary 261

Appendix D Index 269

Welcome

How to use this guide

Civil ET is a node-based design workspace for UK structural and civil engineers. This guide explains the modules, settings and helper tools in version 1.2.0, with screenshots, and answers the practical question "how do I...?" for each task.

Finding what you need

- **Know the task, not the module?** Go to **Appendix A, Quick answers**: short numbered recipes such as *How do I wire a load set to a slab?*
- **Something is not working? Appendix B, Troubleshooting** lists symptoms, likely causes and fixes.
- **Unfamiliar term? Appendix C, Glossary**.
- **Know a key word? Appendix D, Index**, or search this PDF (Ctrl+F). The PDF also has a bookmarks panel listing every chapter and section.
- **New to Civil ET?** Read Chapters 1 and 2, then the chapter for the module you want to use.

Conventions

- **Bold** text is something you see or click on screen: a button, a menu item, a field label or a drop-down value.
- Section references such as *Section 3.4* point to a section in this guide.
- Unit symbols are always lower-case and follow the Eurocode style: mm, kN, kNm, MPa, fyk.
- Screenshots show the default **Dark Navy** theme and the example workspace that Civil ET opens with. Numbers in the screenshots illustrate how the features work; they are not design advice for any real project.
- Modules marked **ALPHA** or **BETA** in the app (Cut & Fill, Sketch-ET, SI map, Base plate) are still being refined; check their results with extra care. Some options are shown but locked in the app because the design is still in development; this guide says so where it matters.

Responsibility. Civil ET is a design aid for qualified engineers. Results must be checked and verified by the responsible engineer. Chapter 15 explains how to check your results.

CHAPTER 1

Introduction and Getting Started

Civil ET is a browser-based structural and civil engineering design workspace. You build a job as a map of "nodes" (a foundation, a column, a beam, a load set, a wind case and so on), wire them together so that loads and sizes flow from one to the next, and read the Eurocode checks, quantities and embodied carbon for the whole job in one place. It is written for practising UK structural engineers and works to the Eurocodes with the UK National Annexes.

This chapter explains what the tool is, what is new in version 1.2.0, how to run it, what you see on first launch, and gives a one-page summary of all 15 node types. Chapter 2 covers the node map in detail. Each node type has its own chapter.

> Civil ET's results are for qualified engineers to verify. The in-app About card says the same ("Design checks to the Eurocodes with the UK National Annexes. Results are for qualified engineers to verify."). Treat it as a design aid and a second pair of eyes, not as a substitute for your own checks and sign-off.

1.1 What Civil ET is

- **A node map.** Every design lives on a card. Cards take their inputs, run the code checks and show utilisations on the card itself. Double-click a card (beam, slab, IFC, cut and fill, wind, Sketch-ET) or use its open button to get the full-screen module for that design.
- **Live wiring.** An OUT dot on one card can be pulled onto an IN dot on another. A load set can feed a slab, the slab can feed a beam, the beam can feed a column, and the column can feed a foundation and a base plate. The data flows live while Auto sync is on.
- **One project, one file.** The whole workspace (node map, wires, every design, project details) is held in the browser and can be saved to a single `.json` file.
- **Reports.** Each module produces a Word-compatible (`.doc`) report in a common layout with live data; the node map can produce a combined report.
- **Carbon.** A live right-hand panel on the node map totals material quantities and embodied carbon for everything on the map (A1 to A5 split, SCORS rating against gross internal area, RIBA-stage trend).

Civil ET runs in your browser. Design data stays in the browser's local storage and in the project files you save. The only things that leave your computer are the optional map, address and elevation look-ups in the wind and Sketch-ET modules, the map and borehole requests made by the SI map, and the optional feedback message.

1.2 What is new in version 1.2.0

The main changes in version 1.2.0 are:

Area	What changed
Sticky note node	A new node type. Typed text, handwriting (Apple Pencil, stylus, finger or mouse), photos, five colours, fold to a small square, and pin to a card or group so the note travels with it.
Example workspace	A browser that has never run Civil ET opens on a worked, wired example map instead of an empty one. It can be reopened at any time from File > "Open the example workspace...".
RIBA stages	Changing RIBA stage now saves the stage you leave. A new stage starts as a copy you can edit or delete from freely; a stage saved earlier reopens as it was.
Live load sets	A load set's values reach every downstream node (slabs, beams, columns, foundations, base plates, maths) as you type, ticks and presets included.
Slab	The cracked long-term deflection feeds the verdict when it is the governing check.
ψ factors	Column and base plate gain an imposed-load category; G is split from F; snow ψ_2 is handled above 1000 m.

Area	What changed
Imposed loads	Categories E13 and E15 to E19 added to the presets.
Sketch-ET	Per-change undo, a foldable ribbon for small screens, a floating address search. Now labelled alpha.
Grid reference card	Accepts a 10/6-figure grid reference, E/N, lat/long and DMS.
Reports	A disclaimer appears in every report.
Look and feel	Theme contrast fixes, column 3D follows the theme, node map text un-bolded, a 2400 kg/m³ concrete take-off density.
Cut and Fill	Now labelled alpha. Base plate is labelled beta.
About card	Updated text and contact details (open it with the `i` button in the header).

Earlier rounds that are still current features include the Eurocode 2 cover/exposure work (BS 8500 exposure class, design working life), the slab FE and Eurocode compliance work, the wind designer's funnelling and shore (OS high-water) work, the global undo manager, touch and pen support, and the node-map performance work. These are covered in the relevant chapters.

1.3 System requirements

Item	Requirement
Browser	A current desktop or tablet browser. Chrome or Edge give the best project-file experience (see below). Safari on iPad is supported by the touch layer.
Computer	A computer with a decent graphics card (GPU). The 3D views, the large node map and the Sketch-ET 3D model all use it, and older or very basic graphics may feel slow on big workspaces.
Internet	A connection is needed to load and use Civil ET.

Project files: Chrome and Edge use the File System Access API, so **Save** writes back to the same file you opened. Other browsers fall back to a download (**Save as...**) and **Open** uses a file picker. Both paths produce the same `.json`.

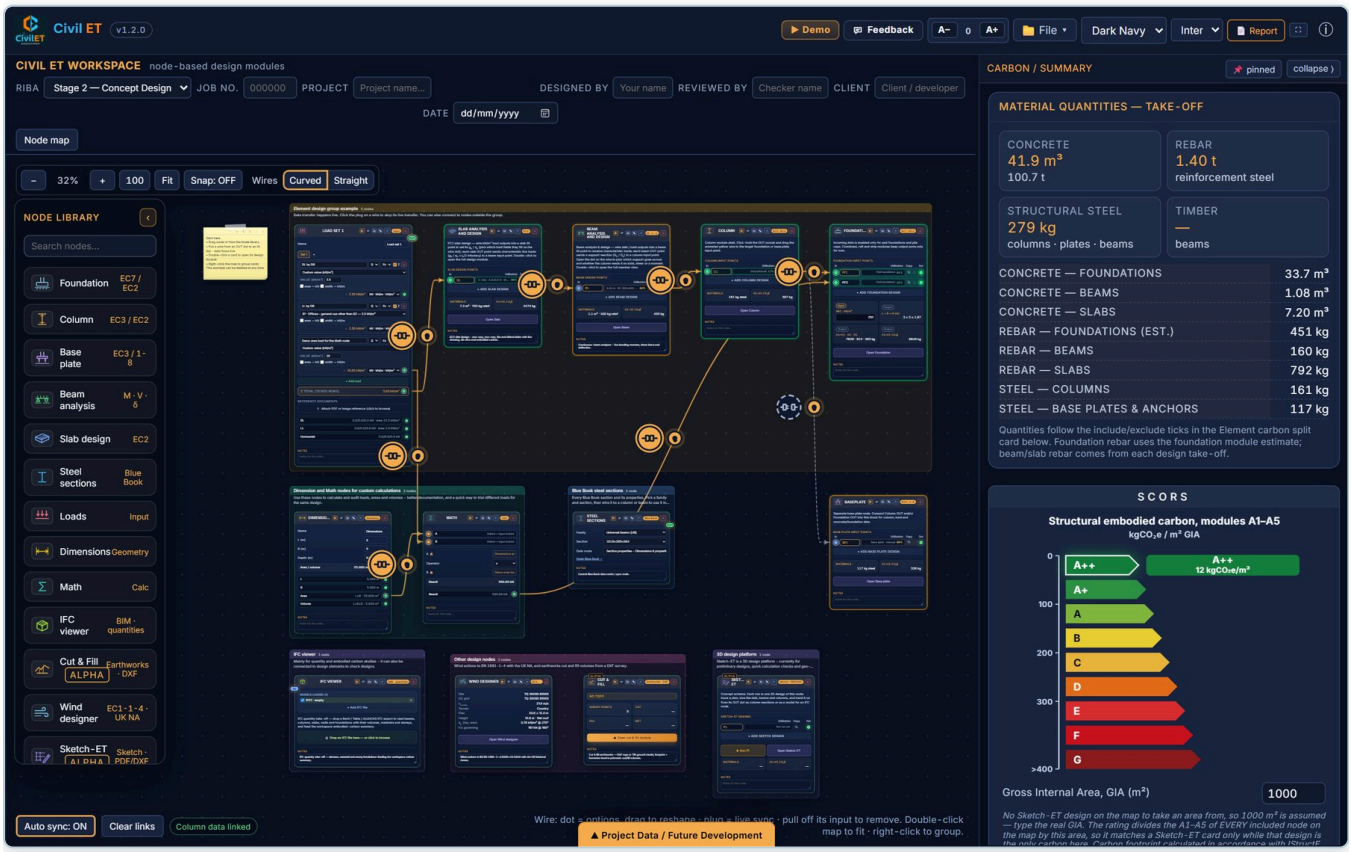
1.4 First launch

1. **The page always opens on the node map.** Civil ET sets the view to the node map on every load, so you never land inside a half-open module.
2. **A brand new browser gets the example workspace.** If no saved Civil ET data is found, a worked example is created for you: a load set feeding a slab, beam and column in a live chain to a foundation and a base plate, the Dimensions and Math nodes working a load into a second foundation, and Blue Book, IFC, wind, cut and fill and Sketch-ET cards, each in a labelled group, plus a sticky note saying where to start. Every design in the example passes. The map is zoomed to fit the first time it is drawn.
3. **Everything auto-saves to the browser.** Each change is written to local storage as you go. A project `.json` file is how you keep, move or share a job (Chapter 2).
4. **Take the guided tour.** The amber **Demo** button in the header (and the small triangle on every card header) walks you through a card, the node library, the title bar, the carbon panel, wiring, and groups and notes.
5. **Fill in the project details.** On the node map the bar at the top holds RIBA stage, job number, project name, designed by, reviewed by, client and date. They stamp every report.

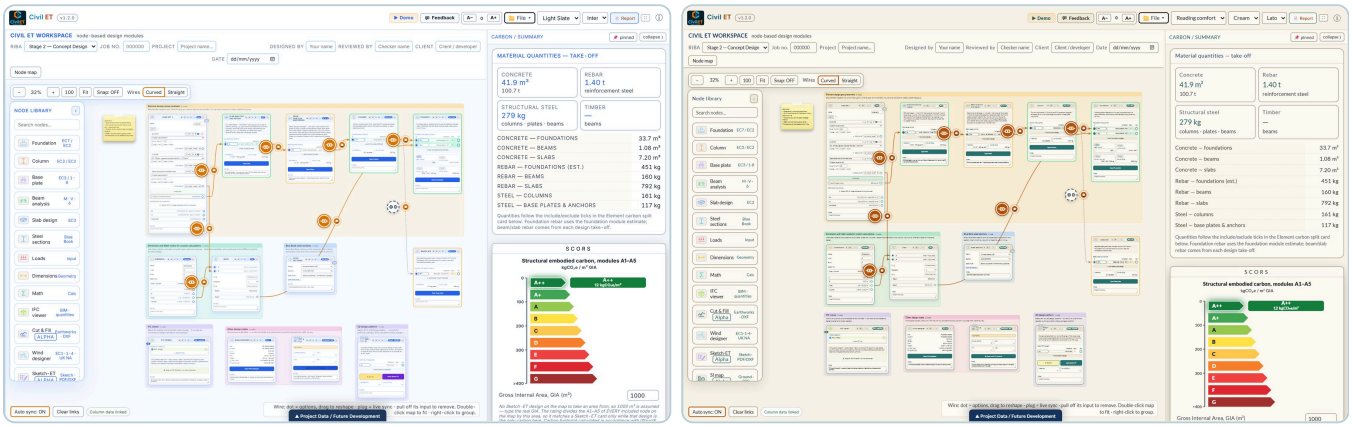
If something goes wrong during start-up, Civil ET shows a "Civil ET failed to start" panel with the error, a **Download backup** button, and a **Clear saved state and reload** button that downloads a backup before it clears anything. See Chapter 2.

1.5 A tour of the screen

The header



Screenshot: The node map on first launch: the node library on the left, the example workspace in the middle, and the carbon and summary panel on the right.



Screenshot: Two of the five colour themes: Light Slate and Reading comfort. Dark Navy, B/W and Warm Sand are also available from the header.

The header runs across the top of every screen. What it shows depends on the screen you are on: the node map or one of the modules below.

Part	Notes
Logo and title	On the node map the title area shows "Civil ET" with the version chip (for example v1.2.0). Inside a module it shows the module name (for example "Beam Analysis & Design") with the code tag (for example "EC2 / EC3") and the version.
Back button	Inside any module a small return-arrow button ("Minimise to blocks") takes you back to the node map. Everything is kept. A **Node map** button in the workspace bar does the same.

Part	Notes
Design picker	Each design module has a picker in the header for its list of designs (foundations PF1, PF2...; columns C1...; beams B1...; slabs S1...; base plates BP1...; and the wind, IFC, cut and fill and Sketch-ET designs).
RIBA stage	A drop-down of RIBA stages 0 to 7 plus "+ Custom stage...". Changing stage saves the stage you leave (Chapter 2).
Job No. and Project name	Edited in place.
Carbon button	Foundation header: toggles between the current foundation's embodied carbon and the all-foundation total.
A- / A+	Scales the large display text. The level runs from -6 to +2.
File menu	New project, Open project, Save, Save as (.json), Duplicate / Copy, Open the example workspace, Project info and metadata, Save stage snapshot, Advance to next RIBA stage, and a list of saved stages to load or delete.
Theme	Dark Navy, Light Slate, B/W, Warm Sand, Reading comfort (with Cream, Pale blue or Pale peach tint).
Font	A long list of UI fonts, with "Aptos / Report style" first.
Report	Opens the report for the module you are in; on the node map it opens the combined report. Disabled while a foundation recalculation is pending.
Info (i) button	The About card: version, a one-line description, the verify-your-results notice, and contact addresses.
Demo	The guided tour.

A "Node link" chip appears in the header of a design module when the design is fed by wires. It says how many wires, which node supplies it, and whether Auto sync is off or the wire is unplugged. Loads that arrive by wire are locked in the input panel because every sync would rewrite them; unplug or delete the wire to edit them by hand.

The modules

The node map is the home screen. A card opens into one of these full-screen modules:

Module	Opens from
Foundation	Foundation card
Column	Column card
Base plate	Base plate card
Beam	Beam card (double-click)
Slab	Slab card (double-click)
IFC model	IFC card (double-click)
Cut & Fill	Cut & Fill card (double-click)
Wind designer	Wind card (double-click)
Sketch-ET	Sketch-ET card (double-click)

The other node types (Steel sections, Loads, Dimensions, Math, Sticky note) are worked entirely on the card.

The input, results and drawer panels

In the foundation module the screen is: an **INPUT** panel on the left (collapsible, resizable), views and calculations in the centre (the split can be dragged, and stacks when the window is narrow), and drawers on the right and bottom. The right drawer and the bottom drawer (soil and site data) can be pinned or float. Other modules have the same idea: inputs to the side, views and checks in the middle, and a context panel on the right.

The ribbon and tool strips

Drawing-based modules (Sketch-ET, Cut and Fill, Wind, IFC) have a tool strip or ribbon across the top of the working area, with open/closed state remembered per module. In Sketch-ET the ribbon can be folded on small screens. Panels in these modules can be opened, closed and reordered, and their widths are remembered.

The right panel on the node map

When you are on the node map, the right panel is the **workspace summary**:

1. Material quantities (take-off) for the whole map: concrete, rebar, structural steel, timber.
2. The SCORS embodied-carbon rating against the project's gross internal area (GIA).
3. Tick boxes to include or exclude each element from both carbon and quantities (one list drives both).
4. Charts of quantities and embodied carbon, and a RIBA-stage trend.

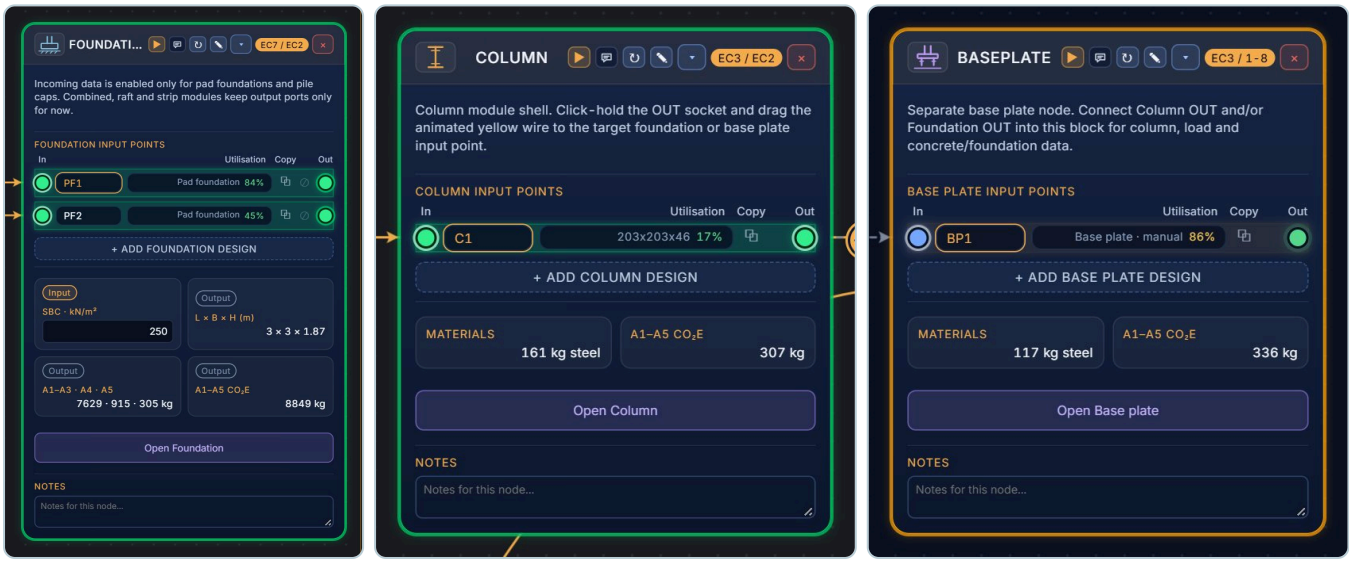
Inside a design module the right panel shows that module's own summary.

Help

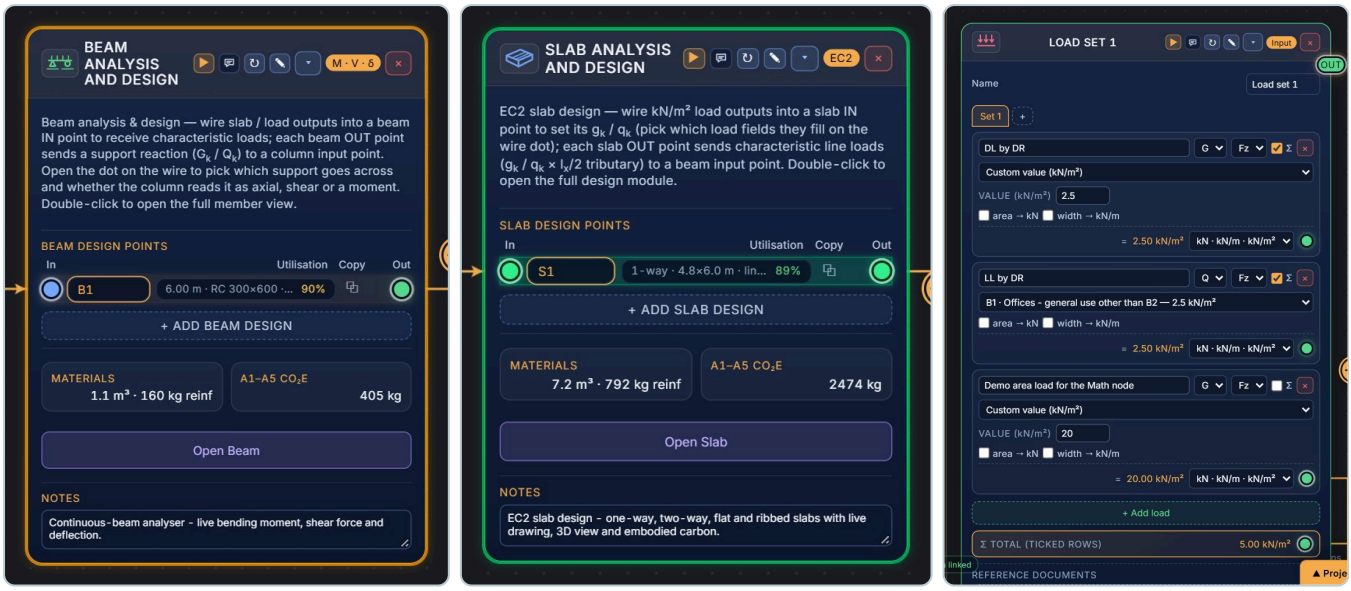
- **Hover tooltips** on almost every button, dot and field.
- **The guided tour** (Demo button and the triangle on each card).
- **Floating "?" help** in the beam module and the other design modules. It opens a small floating box that follows the page theme.
- **A note field at the foot of every card.** The text travels into the report with the design.
- **Feedback buttons**: one for the whole app on the node map and one on each card and module. They are sent to the Civil ET team where the feedback service can be reached, otherwise they open your mail application.

1.6 The 15 node types at a glance

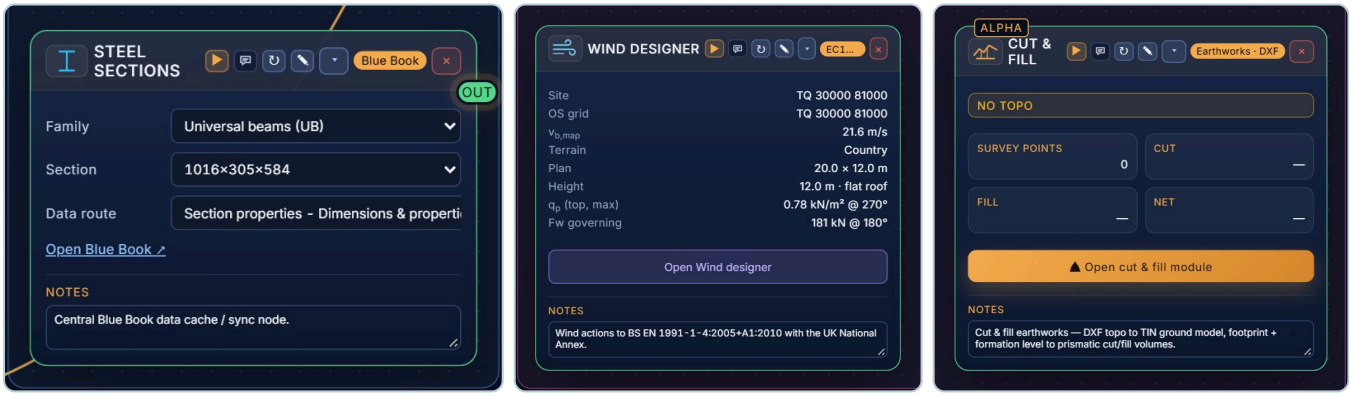
The node library on the left of the node map lists them all. Drag one onto the map, or click it to drop it in the middle of the view. The "Tag" is the chip shown in the library and on the card.



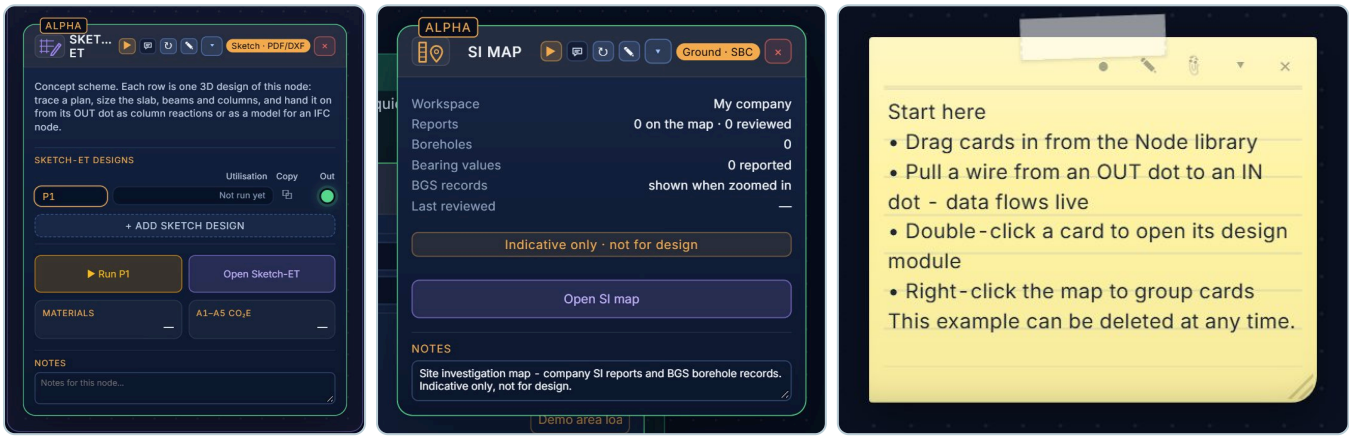
Screenshot: Foundation, Column and Base plate node cards. Each card shows its designs (PF1, C1, BP1), their utilisation and an Open button for the full module.



Screenshot: Beam, Slab and Loads cards.



Screenshot: Steel sections (Blue Book), Wind designer and Cut & Fill cards.



Screenshot: Sketch-ET, SI map and Sticky note cards.

#	Node	Tag	What it is	Standards used
1	Foundation	EC7 / EC2	Pad, combined, strip and pile cap foundations (raft is present but locked in this release). Bearing, sliding, uplift, punching, shear, settlement, crack width, reinforcement and carbon.	EN 1997-1 (Eurocode 7, Design Approach 1), EN 1992-1-1 with the UK NA, NHBC Chapter 4.2 (foundation depth near trees)
2	Column	EC3 / EC2	Steel or reinforced concrete column design.	EN 1993 for steel, EN 1992 for concrete
3	Base plate	EC3 / 1-8	Base plate and anchor design. Beta.	EN 1993-1-8

#	Node	Tag	What it is	Standards used
4	Beam analysis	M, V, δ	Beam analysis (moment, shear, deflection) with member design checks in concrete or steel. Timber is analysis and take-off only in this release.	EN 1992-1-1 with UK NA (cover to BS 8500-1, fire to EN 1992-1-2 tabulated method), EN 1993 steel beam design following SCI P364
5	Slab design	EC2	One-way and two-way slabs, including composite slabs with steel decking.	EN 1992-1-1 with UK NA (including 5.1.3 and 6.4 punching), EN 1992-1-2 Table 5.8
6	Steel sections	Blue Book	Section properties and data from the SCI Blue Book, shared by other nodes.	Blue Book data (EN 1993 section properties)
7	Loads	Input	Load sets: point, line and area loads by class (dead, live, horizontal), with imposed-load category presets.	Imposed-load categories and ψ factors per Eurocode practice
8	Dimensions	Geometry	Simple geometry values (lengths, area) that can be wired onward.	None
9	Math	Calc	Two operands and an operator; used to build a load from dimensions and loads.	None
10	IFC viewer	BIM, quantities	Loads an IFC model, takes off quantities and embodied carbon, and lets chosen elements be designed in other nodes.	IStructE "How to calculate embodied carbon" (3rd ed.) factor tables
11	Cut & Fill	Earthworks, DXF	Earthworks volumes from DXF survey data. Alpha.	None (CRS assumed EPSG:27700 unless set)
12	Wind designer	EC1-1-4, UK NA	Wind actions on a building drawn or imported on a site.	EN 1991-1-4 with the UK National Annex (NA.1 map, OS high-water shore data)
13	Sketch-ET	Sketch, PDF/DXF	Sketch a scheme over a PDF or DXF, size elements at concept stage, and hand reactions on to the detailed nodes. Alpha.	Concept sizing; embodied carbon (SCORS band)
14	SI map	Ground, SBC	Puts site investigation reports (PDF or AGS) and public BGS borehole records on a map, so you can see what is known about the ground. Indicative only, not for design. Alpha.	None (values as written in the reports; BGS records)
15	Sticky note	Notes, ink, photos	Notes, handwriting and pictures on the map. Carries no design data.	None

Notes on the table:

- "Standards used" lists the standards each node names. Each node's own chapter gives the detailed checks and the clauses used.
- Beta and alpha labels: Sketch-ET, Cut & Fill and the SI map are alpha; Base plate is beta. A badge shows beside the name on the card and in the header.
- The Foundation, Column and Base plate nodes are the three "core" cards. If you delete one it is hidden, not destroyed, and can be dragged back from the node library.
- The retired Roadmap node is not offered and is not drawn if an old file contains one.

Chapter 17 lists these standards and data sources with their publishers and web links.

Which wires are allowed

The map only accepts pairs where data actually moves:

From	To
Loads	Beam, Slab, Foundation, Math
Dimensions	Math
Math	Beam, Slab, Foundation, Math
Slab	Beam, Column, Math
Beam	Column, Math
Column	Foundation, Base plate, Math
Foundation	Base plate, Math
Base plate	Foundation, Math
Steel sections (Blue Book)	Column (the section is read there), Base plate, Foundation, Math (wires allowed, not read)
IFC	Beam, Slab, Column, Foundation, Math
Wind	Beam, Slab, Column, Foundation, Base plate, Math
Sketch-ET	Column, Foundation, Base plate, Beam, Slab, Math, IFC (the IFC link is two-way)

The SI map and sticky notes never take part in wiring. Chapter 2 explains ports and how a wire is refused (for example an area load wired into a column input).

1.7 Tips and known caveats

- **Work is stored per browser.** Clearing site data deletes it. Save project files regularly and keep them in your document system.
- **Opening a project reloads the page.** The node map and modules read their state at start-up, so Open, loading a saved RIBA stage and "Open the example workspace" all end with a reload.
- **Large IFC models.** The model's 3D solids are kept in the browser if they fit; when storage is nearly full they are kept for the session only and the saved card reopens with bounding boxes until you load the IFC file again.
- **Foundation recalculation is deliberate.** The foundation calculation waits until a foundation card is on the map or the foundation module is opened. A design is not recalculated just because you zoomed, added another card or reopened it; only a change to its numbers does that.
- **Raft foundations** appear in the type list but are disabled ("in development, locked").

CHAPTER 2

The Workspace and Node Map

The node map is the home screen of Civil ET. This chapter covers how to build a map (adding, connecting, grouping and deleting nodes), how wires and ports behave, how to move around the map by mouse, trackpad, finger or pen, how undo and redo work, how a job is saved, loaded and moved between browsers, and the project-wide features that sit around the map: the example workspace, sticky notes, RIBA stages, live load sets, project details, the carbon panel, performance behaviour and the start-up crash guard.

2.1 Overview

The workspace is a canvas of cards (nodes). Each card is a view of a design: a load set, a slab, a beam, a column, a foundation and so on. Wires carry data from the OUT side of one card to the IN side of another. The canvas has:

- a **node library** on the left (search, drag or click to add);
- the **cards**, wires and optional **groups** in the middle;
- a **zoom strip** at the bottom (`-`, readout, `+`, `100`, `Fit`, Snap, wire style);
- an **Auto sync / Clear links** strip with a status line;
- the **project bar** across the top (RIBA stage, job number, project, designed by, reviewed by, client, date) and the **Node map** button;
- the **workspace summary** (quantities and carbon) in the right panel.

A one-line reminder is printed on the canvas: "Wire: dot = options, drag to reshape - plug = live sync - pull off its input to remove. Double-click map to fit - right-click to group."

2.2 Adding nodes

There are three ways to add a node.

1. **Click** an entry in the node library. The card appears near the middle of what is on screen, nudged a little for each new card so a few clicks do not stack them exactly.
2. **Drag** an entry from the library onto the map. The card lands where you drop it. If you drop it on a group's frame it joins that group.
3. **Drop an `.ifc` file** anywhere on the map. A new IFC viewer node is created at that point and the file is loaded into it. Any other file dropped on the map is swallowed with a warning ("Only .ifc files can be dropped on the node map") so the browser never navigates away from the workspace.

The library has a search box that filters by name, tag or node type. It can be collapsed with the arrow in its heading, and it remembers that state.

Things to know when adding:

- **Foundation, Column and Base plate are "core" cards.** They exist on a fresh map. Deleting one hides it rather than removing it; a fresh copy can be dragged back from the library. If a wire needs a hidden core card as an endpoint, the card is brought back automatically and the status line tells you.
- **A card adds no design by itself.** For foundation, column, base plate, beam and slab, every card of a kind is a view of one workspace design list. Dropping a second beam card does not create a second beam design; use the "+ Add" button on the card when you want a new design. The first card of a kind added to a map that has none starts that design list again at one design (the status line says so).
- **Only the first IFC card is drawn.** Further IFC models are folded into the first card's model list, so "node added" for a second model refers to a row in that card rather than a new card.
- **Library entries marked under development are locked.** None are locked in 1.2.0.
- **Beta/alpha badges** appear next to Sketch-ET (alpha), Cut & Fill (alpha), SI map (alpha) and Base plate (beta).

Card controls

Every card header carries the same small controls (some nodes omit some).

Control	Does
Triangle (tour)	Guided tour: this card, then the workspace around it.
Speech bubble	Feedback about this card (Section 2.14).
↻	Sync this node and everything wired to it now, even if its wires are paused.
✎	Rename the title in place. Double-clicking the title does the same; Enter commits, Escape cancels.
▾ / ▸	Minimise the card to its header (output dots stay) or expand it.
Tag chip	The code the card designs to (for example EC3 or EC1-1-4).
×	Delete the node (Section 2.5).

Beam and slab cards keep fixed titles ("Beam analysis and design", "Slab analysis and design"). Every design card has a notes field at the bottom; the text is carried into the report with the design.

2.3 Ports, wires and links

Sockets

Cards expose two kinds of socket.

- A **main OUT socket** (and, on the IFC card only, a main IN socket). Used by the core cards, Load sets and Steel sections. Hover text: "Main output socket - drag to connect; release in empty space to delink latest wire" and "Input socket - hold and pull away to delink the latest incoming wire".
- **Per-design or per-row dots** on the card body: an IN dot on the left and an OUT dot on the right of each design row. Beam, Slab, Math, Dimensions, IFC, Cut & Fill and Sketch-ET cards use only these dots (no main OUT), because a main socket would not say which design, operand or dimension is meant. The Wind card has no main OUT socket either; its wires are drawn from its own dots.

Where a wire lands inside a card is part of the wire:

- A wire into a **foundation** or **base plate** lands on "all designs" or on one named design (for example PF2), chosen per wire in the wire's options. Several column outputs can therefore feed different foundation designs at the same time.
- A wire into a **Math** node lands on operand A or operand B, and each operand takes one wire only. A second wire to an occupied operand is refused ("delink it first (one wire per input)").
- A wire from an **IFC** element to a design is one wire per element you chose to design.

Which pairs can be connected

A wire is only accepted for a pair where data actually moves. The table is reproduced from Chapter 1 for convenience.

From	To
Loads	Beam, Slab, Foundation, Math
Dimensions	Math
Math	Beam, Slab, Foundation, Math
Slab	Beam, Column, Math
Beam	Column, Math
Column	Foundation, Base plate, Math

From	To
Foundation	Base plate, Math
Base plate	Foundation, Math
Steel sections	Column, Base plate, Foundation, Math
IFC	Beam, Slab, Column, Foundation, Math
Wind	Beam, Slab, Column, Foundation, Base plate, Math
Sketch-ET	Column, Foundation, Base plate, Beam, Slab, Math, IFC

Anything else is refused with "Invalid link for this node pair". Sticky notes and the SI map take no wires. A node cannot connect to itself. The wire label shown on a link names the pair (for example "SLAB LOADS -> BEAM", "BEAM REACTIONS -> COLUMN", "COLUMN DATA -> FOUNDATION"). Sketch-ET to IFC is the one two-way exchange: the scheme goes across as a concept IFC model and the embodied-carbon factors chosen in the IFC node come back to move the Sketch-ET carbon band. It is drawn with arrowheads at both ends and stored as a single wire.

Unit checks

Before a wire is made, Civil ET checks, where the source declares it, that the target can use the unit being carried. A refused wire does nothing else; it does not remove other wires on its way out.

Target input	Accepts
Beam	Point loads (kN), line loads (kN/m), area loads (kN/m^2)
Slab	Area loads (kN/m^2) only
Column	Point loads (kN) only
Foundation	Point loads (kN) only
Base plate	Point loads (kN) only

A slab's area load that carries its own loaded width is passed to a beam (which sheds it over the width) or a column (which takes a quarter panel); other targets refuse it. When the source does not declare a unit (a beam's reactions, a column's output, the Steel sections node) the wire is never refused for unit reasons.

One source per design

If you drop a wire on a design that was being driven from another card, the older wire is removed and the status line says so ("... It is now designed from ..."). A wire is persistent data: which design it feeds does not depend on which design tab you have open.

Making, reshaping and removing wires

- **Connect:** drag from an OUT socket or dot onto an IN socket or dot.
- **Remove:** pull the wire off its input; or release a drag from an OUT socket in empty space to delink the latest wire; or open the wire's dot menu and choose Delete connection.
- **Reshape:** drag the dot in the middle of a wire to bend it (curved wires) or to move its right-angle bend (straight wires). The wire style switch at the bottom of the map chooses between **Curved** (bezier) and **Straight** (right-angle bends). The choice is remembered.
- **Options:** click the dot for the wire's menu. For a column-to-foundation wire this is the "Connection transfer options" box: which foundation design receives the data, which fields are transferred, a preview table, "Apply / check transfer" and "Delete connection". The base plate has its own "Base plate load transfer" box.
- **Clear links** removes every wire at once and also removes the locked, transferred loads from the beam designs.

Live sync

Syncing is controlled at three levels.

Level	Control	Effect
Whole map	**Auto sync: ON / OFF** button	Master switch. Default is ON (stored). When OFF, new wires connect but do not carry data until you press Sync.
One wire	The **plug** beside the wire's dot	Pauses or resumes that wire's live sync. A paused wire keeps the data of its last sync. Wires saved before the plug existed are live.
One node	The card's ↻	Pushes now, even through a paused wire or with Auto sync off.

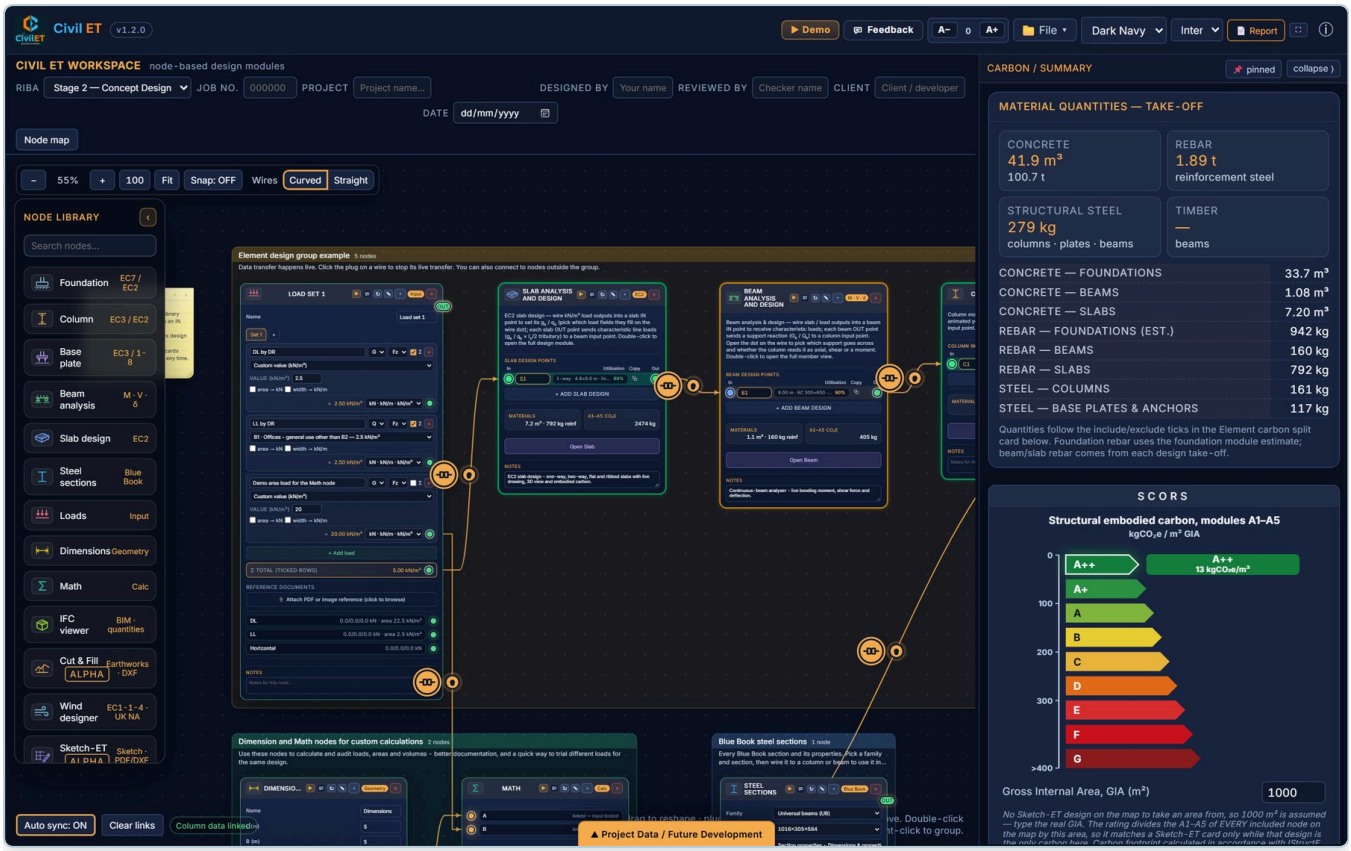
A wire updates itself only while both Auto sync and its own plug are on. Typing in a card field does not rebuild the map under your cursor; the full sync runs when focus leaves the field. Load sets are the exception (Section 2.12).

Data that arrives by wire is **locked** in the receiving design's input panel, because every sync rewrites it. A "Node link" chip in the module header says which node feeds it and whether the wire is live, unplugged or waiting on a manual sync. Unplug or delete the wire to edit those values by hand.

A completed design is not recalculated just because you opened it, zoomed, or added another card. Each wire remembers its last delivery and a transfer that carries the same numbers (ignoring time stamps) is a no-op; only a changed number causes a redesign. A manual ↻ always sends.

2.4 Selecting, moving, aligning and grouping

- **Move a card** by dragging its header. **Snap** (button in the zoom strip, off or on) lines a dragged card or selection up with the top, bottom, left, right edges and centres of nearby cards, showing dashed guides, and otherwise lands it on a 20 px grid. It affects dragging only.
- **Select several** by dragging a box across empty map (left button), or shift-drag. A selection bar appears with **Delete selected** and **Clear**. Escape clears the selection.
- **Right-click** the map or a card for a menu: "Group selected nodes...", "Align tops of selected" (two or more), "Remove from group", and for a group: Rename, Add or Edit comment, Select the group's nodes, Ungroup. The View section repeats Zoom to fit and the Snap switch.
- **Groups** are a rounded translucent frame behind their cards with a title bar. A group has a name, a comment and a colour; it moves as one and keeps related designs together. A node can be connected to nodes outside its group. Groups are saved with the map and are covered by undo.
- **Delete** with the × on a card, or select and press Delete or Backspace (not while typing in a box). Deleting several nodes asks once and rebuilds the map once.



Screenshot: A group on the node map: a load set feeds a slab, the slab feeds a beam, the beam feeds a column and the column feeds the foundations. The group frame can be moved as one.

2.5 Deleting nodes - what else goes

- All wires attached to the node are removed.
- Deleting a **core** card hides it; its wires are dropped.
- Deleting a **beam or slab** card releases any IFC element designs that wrote load bands into the beam or fields into the slab, the same way as pulling each wire would.
- Deleting an **IFC** card closes its viewer, releases its element designs and frees its stored geometry, unless a saved RIBA stage still holds that card.
- Deleting a **note** frees its pictures unless another note shares them.
- Undo (Section 2.8) brings a deleted node back with its design, not as a blank card.

2.6 Moving around the map

Mouse and trackpad

Action	Result
Wheel notch	Zoom by 10 % per notch, smoothly, around the pointer.
Trackpad scroll / pinch	Zooms in proportion to the gesture (pinch is faster). Safari on a Mac uses gesture events for pinch.
Middle-button drag	Pan the map.
Left-drag on empty map	Selection box (not pan).
Double-click empty map	Zoom to fit every node.

Action	Result
`-` `+` buttons	Zoom out or in by 25 %.
`100`	Reset zoom to 100 %.
`Fit`	Zoom to fit every node (never above 100 %).

The zoom range is 1 % to 220 %. The view (position and zoom) is saved when it settles.

Touch and pen (iPad, Android and Windows tablets)

There is no tablet mode switch. A finger or an Apple Pencil works wherever it lands, alongside the mouse.

Gesture	Result
One finger on a card or control	Left mouse button (tap = click, double-tap = double-click).
One finger on empty map	Pan.
Two fingers	Pinch-zoom and pan together on the node map. In other surfaces, a pinch becomes the wheel and two fingers moving together become the middle button.
Hold about half a second	Right-click (context menus: groups, beam supports and so on).
Hold, then drag on the map	Selection sweep.
Hold and lift on empty map	The map menu.
Double-tap empty map	Zoom to fit.

Scrolling panels, text boxes, selects and plain buttons stay with the browser. Wire dots, ports and splitters are given a larger touch target. Library entries and reorderable panels are picked up by holding them or sliding sideways. Sticky notes treat a pen as always writing; a finger or mouse writes once the pencil toggle is on (Section 2.11).

2.7 The right panel on the node map

The right panel shows the workspace summary:

1. **Material quantities** (concrete, rebar, structural steel, timber, with a take-off table).
2. **SCORS** embodied-carbon rating against the gross internal area (GIA).
3. **Quantity override ticks** and an **Element carbon split** with an include/exclude tick per element. The same list drives both the carbon totals and the quantities.
4. **Quantity charts** and **embodied CO_2e charts**.
5. A **RIBA-stage trend**.

The GIA is one workspace figure. A value you type wins; otherwise the area is derived from the included Sketch-ET designs; with nothing to derive from, 1000 m^2 is used and the panel says so. The GIA also feeds the IFC take-off's A5.2 allowance, so changing it moves those figures. With nothing on the map, every total stays at zero until a design node is added.

2.8 Undo and redo

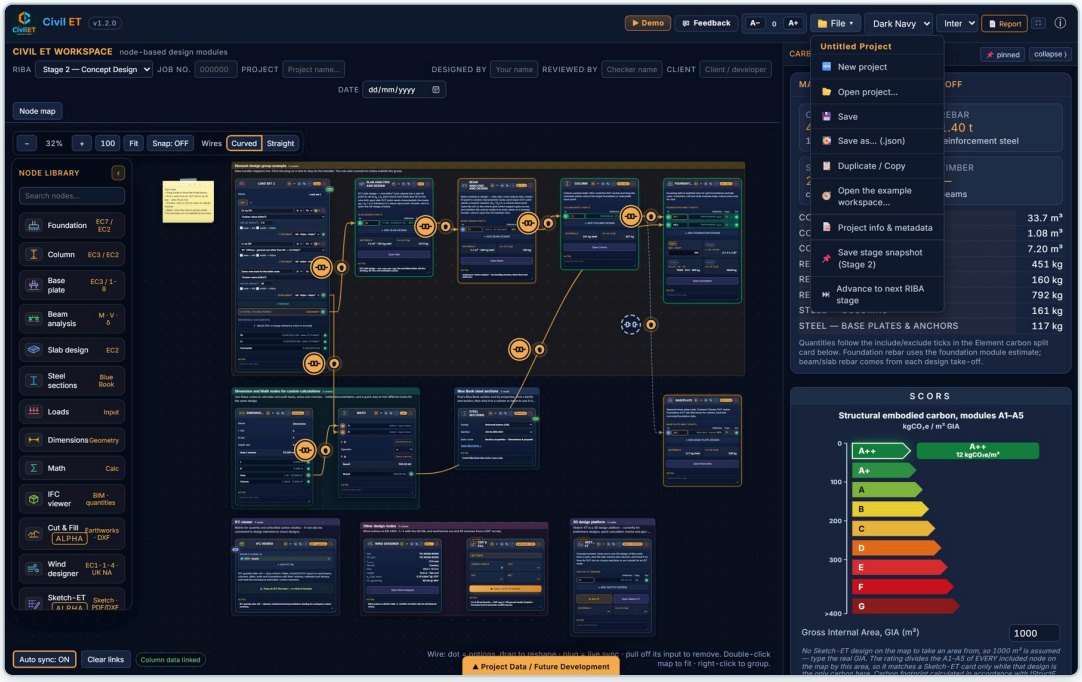
Undo is global and works by snapshot, not by recording each edit.

Item	Detail
Keys	`Ctrl+Z` undo; `Ctrl+Y` or `Ctrl+Shift+Z` redo (`Cmd` on a Mac).
History	One history per design, 80 steps deep each: the node map (nodes, positions, wires, core visibility, collapsed state, groups, titles and the foundation designs), each foundation set, each column, base plate, beam and slab design, each wind case and each IFC model.
How steps form	Civil ET snapshots the active design just before each interaction and again a moment (about 0.3 s) after it. If the snapshot changed, the earlier one becomes an undo step. Keystrokes in one box within about 1.2 s are a single step.
What is restored	The **inputs**. The module then recomputes, so no stored result can go stale.
Feedback	A small toast says "Undo - Beam B1 - 3 more", "Nothing to undo" and so on.
Not covered	Cut & Fill and Sketch-ET keep their own drawing-level undo and redo; the global keys defer to them. Free-text boxes (notes and dialogs) keep the browser's own text undo. Undo is ignored while a dialog is open.
Lifetime	History is in memory. It is lost on reload.

A box being typed in may hold a value it has not handed to the design yet. Undo commits it first (it blurs the box) so the step takes back exactly what is on screen.

2.9 Saving, loading and exporting

Civil ET works in two layers.



Screenshot: The File menu: New project, Open project, Save, Save as, Duplicate / Copy, Open the example workspace, Project info and metadata, Save stage snapshot and Advance to next RIBA stage.

Layer 1: automatic browser storage

Everything is written to your browser's local storage as you work; nothing needs saving to carry on tomorrow in the same browser.

The browser also holds the remembered file handle, the project's fee document and the stored IFC geometry.

Local storage is per browser and per site address; it is lost if site data is cleared and is not shared between machines. It has a quota. When it is exceeded Civil ET degrades gracefully: saved RIBA stage snapshots drop their workspace copy from the browser (the project file still carries them), a note picture is refused, and large IFC solids are kept for the session only.

Layer 2: project files (`.json`)

The **File** menu in the header:

Item	What it does
New project	Confirms, then resets the foundation designs, project details and related settings to defaults and forgets the file handle. It is a reset of the foundation side; delete or clear the node map separately if you want an empty map.
Open project...	Picks a `.json` (also `.padfound.json`, the older name). Parses it, stages the state, writes the workspace keys and **reloads the page**, because the modules read their state only at start-up.
Save	In Chrome/Edge writes back to the file you opened or last saved (the handle is remembered across the reload). Elsewhere it stores a copy in browser storage and tells you to use Save as for a disk file.
Save as... (.json)	Downloads the project.
Duplicate / Copy	Downloads a copy with " (Copy)" added to the project name and today's date.
Open the example workspace...	Replaces the map and every design in this browser with the example (Section 2.10). Asks first.
Project info & metadata	A form for project name, number, designed by, reviewed by, client, date, RIBA stage and notes, with the list of saved stages.
Save stage snapshot / Advance to next RIBA stage / Load stage	See Section 2.13.

The file name is built as `<RIBA stage>_<job number>_<project name>_v<version>.json`; empty parts drop out, an em dash in a stage label is removed, and "Untitled" stands in for a missing project name.

A project file contains: project details and saved stages, foundations, NHBC state, module switches, custom EPD and steel factor sets, card and result orders, output toggles, the global fee document setting and the **whole workspace** - everything the app stores in the browser except the steel-section data, which it can rebuild. Opening a project replaces the workspace rather than merging, so nodes deleted before the save do not come back from whatever the browser still held. A file from before the workspace was included loads the foundation data only and leaves the node map as it is. Old files with shared Mx/My moments are migrated to per-load-case moments on opening.

Reports

The header **Report** button opens the report for the module you are in; on the node map it opens the combined report. Reports are Word-compatible `.doc` files in a common layout (cover and status block, then numbered chapters on new pages: project details, summary of checks, views and design diagrams, working calculations, sustainability, references). They follow the actual design, show live data and carry the application version and a disclaimer. Each module chapter says what its report contains.

2.10 The example workspace

On the very first run in a browser (no stored Civil ET data), the app opens on a ready-made map instead of an empty one, so the idea of nodes is obvious:

- a load set feeding a slab, beam and column chain, then a foundation and a base plate, all wired live;
- Dimensions and Math nodes working a load into a second foundation;
- Steel sections, IFC, wind, cut and fill and Sketch-ET cards;

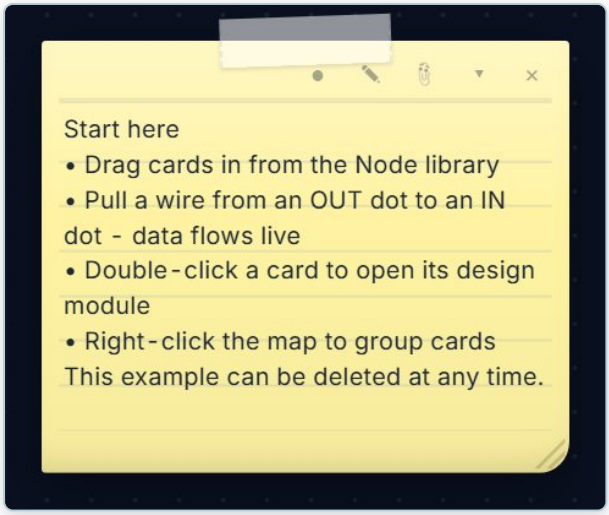
- labelled groups (the first is named "Element design group example" and its comment explains that data transfers live and that the plug on a wire stops its live transfer);
- a sticky note saying where to start.

Every design in the example passes. The map is zoomed to fit the first time it is drawn.

To get it back at any time use **File > Open the example workspace...**. It replaces the node map and every design in this browser (your project details and saved stages are kept), so save your project to a file first if you want to keep it.

2.11 Sticky notes

The Sticky note node (tag "Notes - ink - photos") puts notes on the map. It is an ordinary node for positioning, zooming, selecting, grouping, deleting and saving, but it carries no data and takes no wires.



Screenshot: A sticky note on the node map. Notes carry text, ink or photos and take no part in wiring.

Feature	How it works
Text	Type straight onto the paper (placeholder "Type a note..."). Saved on a short delay without redrawing the map.
Ink	An Apple Pencil or stylus always writes. A finger or the mouse writes once the pencil toggle is on. Strokes are stored as vectors in the note's own pixel space (pressure sets width), so they stay sharp at any zoom. Three ink colours; undo-last-stroke and clear-all buttons. Past about 150,000 numbers the oldest strokes are dropped.
Pictures	Paperclip button (an iPad offers the camera), paste, or drop an image on the paper. Each is scaled to 1024 px JPEG and stored under its own key so a big picture cannot stop the map from saving. Up to 6 pictures per note and a shared budget of about 2.6 million characters across the project. Click a picture to enlarge it.
Colour	Yellow, pink, green, blue or orange.
Fold	▾ folds the note into a small dog-eared square; a click opens it. The square shows icons for text, ink and picture count.
Size	Drag the corner grip. Default 240×170.
Pin	Let go of a note over a card and it sticks to it (a pin replaces the tape) and travels with that card. Over a group frame it joins the group. On empty map it is free again. The pin button unsticks it.

Pictures travel in the project file because they are stored with the workspace. If the browser's storage is full a picture is refused with a message.

2.12 Live load sets

The Loads node holds one or more **load data sets** (tabs, like worksheet tabs). Add a tab with `+`, rename by double-click or the pencil, delete with `×` (click twice to confirm). Rows carry point, line or area loads by class (dead, live, horizontal) in kN, kN/m, kN/m^2 or kN/m^3, with imposed-load category presets (for example A1 to A7 and further categories, including E13 and E15 to E19 added in 1.2.0).

Since 1.2.0 a load set updates **every card downstream of it** as you type. Presets, class changes, the area and width ticks, and adding or deleting a row are treated as finished edits and sync at once. While you type, the cards reached through the wires (slabs, beams, columns, foundations, base plates, maths) are brought up to date on a short throttle and rebuilt in place; the card you are typing in is never touched, so focus and caret stay put. A slab fed this way passes its new area loads on to the beams it carries. This requires Auto sync on, a live wire and the node map open.

2.13 Project details and RIBA stages

Project and global settings



Screenshot: The workspace bar: RIBA stage selector and the project detail fields that stamp every report.

Setting	Where	Notes
Project name, job number	Project bar; File > Project info	Appear on reports and in the saved file name.
Designed by, reviewed by, client, date	Project bar; File > Project info	Report cover details.
Notes	File > Project info	Free text.
RIBA stage	Project bar or header drop-down	Stages 0 to 7 (Strategic Definition, Preparation & Briefing, Concept Design, Spatial Coordination, Technical Design, Manufacturing & Construction, Handover, Use) plus custom stages. Default for a new project is Stage 2.
GIA	Right panel	One figure for the SCORS rating (Section 2.7).
Theme and font	Header	Five themes and a long font list.
A- / A+	Header	Scales large display text (-6 to +2).
Auto sync, Snap, wire style	Node map	Remembered.

Custom stages: choose "+ Custom stage..." and enter text such as "Stage 4.5 - Tender / Contractor Design". A bare number is expanded to "Stage n - Custom Stage". Custom stages are saved with the project.

What a stage switch does

RIBA stage control from the header, the project bar, the project-info form and the File menu all go through the same routine, so they behave identically.

1. The stage you are **leaving is saved** in full (node map, wires, every design module, foundations, NHBC state, module switches, and the whole-map carbon split).
2. If the target stage **already has a snapshot**, it is opened from that snapshot (a reload, as for opening a project file). Your changes in one stage never reach another.

3. If the target stage has **no snapshot yet**, it starts as a **copy** of the one you just saved: the live design carries on under the new name. From then on, edits and deletions belong to the new stage only.

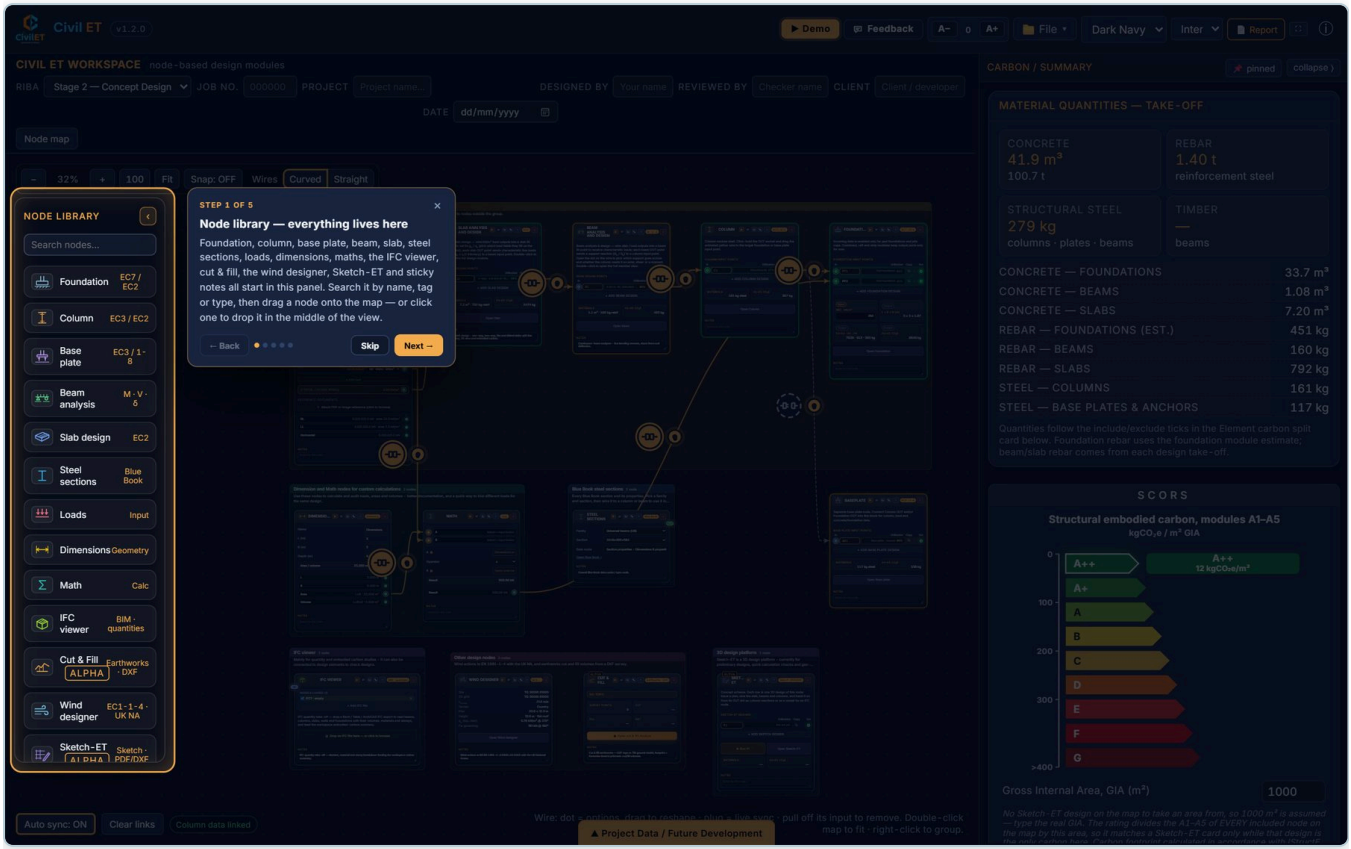
The stage controls in the File menu are:

- **Save stage snapshot (Stage n)**: stores the current stage's snapshot in the session. Use "Save as" to persist it to disk with the project.
- **Advance to next RIBA stage**: switches to the next stage in the list (and says "Already at final RIBA stage" at the end).
- **Load stage**: lists saved stages, each with the date and a delete cross. Loading the stage you are already in asks before discarding changes made since it was saved.

The stage bar shows a tick and a count for the number of saved stages. The RIBA trend in the carbon panel prices past stages from the whole-map A1 to A5 split stored with each snapshot.

2.14 Feedback, help and the tour

- **Feedback** buttons exist for the whole app (node map), each card and each module. Sending delivers your message to the Civil ET team. If it cannot be delivered, the entry is kept in the browser, the dialog offers Copy and Download .json, and the browser opens an e-mail to the team addresses.
- **Guided tour**: the Demo button and each card's triangle.
- **About**: the `i` button in the header.



Screenshot: Step 1 of the guided tour, highlighting the node library. Back, Skip and Next move through the tour.

2.15 Performance and level of detail

Large maps are designed to stay smooth.

- **Level of detail.** Below 30 % zoom, each card is drawn as a tile (frame, title bar and connection dots) and the browser skips its contents.
- **Off-screen cards.** Cards more than half a screen outside the view are not drawn at all, but their wires are.

- **Smooth view.** Zooming and panning stay smooth, and the view is saved when you stop moving.
- **No needless work.** An idle map does almost nothing. Searches for alternative sections run only when you press their button, and their results are kept while the inputs are unchanged.
- **Wire look.** A live wire is a solid colour; a paused one is dashed.

If a map still feels slow, collapse cards you are not using (▾), zoom out to the tile level, or split the job into RIBA stages or files.

2.16 Start-up crash guard and backups

If something goes wrong while the app starts - most likely because saved data from an older build does not suit the current one - Civil ET does not leave a blank page. It shows a panel headed **Civil ET failed to start** with the error text and three buttons:

Button	Does
Download backup	Saves every key in the browser's local storage for this site to `civil-et-saved-data-backup.json`.
Clear saved state and reload	Downloads the same backup first, then clears local storage and reloads.
Dismiss	Closes the panel.

Send the backup and the error text to the support addresses on the About card if you need help recovering a job. The backup file is not a project file; recovery is by restoring the keys, so keep project `.json` files as your real archive.

2.17 Tips and limitations

- Save a project file before using Open, Open the example workspace, a stage switch or New project - several of them replace what is in the browser or reload the page.
- Two browser tabs on the same site share one local storage. Work in one tab at a time.
- A map saved by an older build is migrated by the modules on load, as for any saved project.
- Optional cards you do not use cost almost nothing, but heavy cards (IFC, wind, Sketch-ET) hold geometry in memory; delete them when finished.
- Undo is per design. After switching module, undo acts on that module's design, not the map.
- Wires run between cards of the same map. There is no link between separate project files.

CHAPTER 3

Foundation designer

3.1 Overview

The Foundation designer is the original Civil ET module. It designs one foundation at a time to Eurocode 7 (geotechnical checks, Design Approach 1) and Eurocode 2 (concrete design), using the UK National Annexes. It keeps a list of foundation designs (PF1, PF2 and so on) and shows the active one in a three-column layout: inputs on the left, views (plan, section, 3D) in the middle, calculations and results on the right.

It handles five foundation systems:

System	Selector label	Status
Pad	Pad foundation (PF)	Full design. The only type that takes live column data and the only type with the alternative-designs search and the NHBC tree check built around it
Combined footing	Combined footing (CF)	Full design, any number of columns on one base
Strip footing	Strip footing (SF)	Full design, per metre run of wall; can be reinforced or plain concrete
Pile cap	Pile cap (PC)	2, 3, 4, 5, 6, 7 or 8 piles, with EC7 pile checks and EC2 cap design
Raft	Raft foundation (RF)	Listed in the selector but locked: the option reads "in development, locked" and cannot be chosen

Every design reports embodied carbon (A1 to A5), and the module produces a detailed or simple calculation note (see the Reports chapter).

Read this chapter alongside the Reports chapter (how the note is built), the Column and Base plate chapters (what feeds a foundation) and the Node map chapter (how the Foundation node card behaves).

3.2 Opening the designer

There are two ways in.

1. **From the node map.** Add a Foundation node from the node library (the card is titled "Foundation analysis design", tag "EC7 / EC2"). Double-click the card to open the designer. The card lists one input point per design (PF1, PF2 ...) with the foundation type and the worst utilisation as a percentage, coloured green, amber (above 85 %) or red (above 100 %).
2. **From the design list.** Each design listed on the Foundation card (PF1, PF2 ...) is an input point; opening a design from the card brings up the designer on that foundation. The same designer is the screen on which the header Report button produces the foundation note described below.

New designs are added from the Foundation node card, which copies the active design's inputs.

Screen layout

Area	Contents
Header	Foundation-system selector (Pad / Combined / Strip / Raft (locked) / Pile cap), pile count selector for pile caps, the "Use column node" toggle, theme and font choices, project file menu, **Report** button
Left, INPUT	Cards: Bearing Capacity, Materials, the load cards, Surcharge, Punching Shear & Calculation Options, Reinforcement. Cards collapse; the load cards can be dragged into a new order

Area	Contents
Middle, views	Two cards, each with a drop-down for Plan View, Section View or 3D Viewer. The "B/W detail" button on the plan card swaps the coloured plan for the black-and-white detail drawing
Right, calculations	Summary table and one card per check (Bearing Pressure, Punching, Bending Reinforcement, Top Reinforcement, Sliding/Overturning/Uplift, Settlement, Alternative designs). Each card has an expandable Working
Bottom drawer	Soil Report, NHBC / Trees, Reference, Project Data, Future Development

The Report button is disabled while a calculation is pending or has failed, and an open report window closes if that happens.

3.3 Foundation types and how they are modelled

Pad (PF)

A rectangular pad under one column, with the column position (eccentricity from the pad centre) free. The geotechnical checks and the EC2 design are described in detail below. This is the only type the node map can drive with live column data.

Combined footing (CF)

A rectangular base carrying two or more columns. The columns have their own size, position and G/Q/W loads, and can be locked to the base (the "column position lock", on by default) so that resizing the base moves the columns with it. Dimension chains between columns can be edited and individual dimensions locked. The tool ships five presets (named after the reference cases they came from) and starts from "Chapel double column" when a pad is converted.

Design basis:

- Soil pressure is the rigid-base linear pressure (the simple method). A stiffness test (Hetenyi, with the subgrade modulus from Vesic, or a typed value) runs as a check and classes the base as rigid, flexible or unchecked; a two-way plate model also runs as an automatic check.
- Actions follow the same Design Approach 1 sets as the pad, applied per column and per component (see "Action basis").
- Alternative designs are searched on request, as for the pad.

Strip footing (SF)

A continuous wall footing designed per metre run across its width. Inputs are wall width, an eccentricity of the wall on the strip, and wall loads per metre (G, Q, W vertical, horizontal and moment). The default is an unreinforced strip, checked as plain concrete by the EC2 12.9.3 projection check. Switching "Reinforcement design" on gives a reinforced strip with flexure, minimum steel, one-way shear and crack width. Defaults: 1.0 m by 0.6 m, 1.0 m thick, C28/35, SBC 123 kN/m².

Pile cap (PC)

Templates for 2, 3, 4, 5, 6, 7 and 8 piles. The 3-pile cap can be triangular (strut-and-tie) or rectangular (beam theory). Piles are circular concrete piles. Defaults: pile diameter 0.45 m, spacing 1.35 m ($3\times$ diameter), edge clearance 0.15 m, embedment into the cap 0.10 m, pile length 10 m, positional tolerance 75 mm, allowable pile compression 1400 kN, tension 0 kN, lateral 75 kN (a second set of allowables is used for Combination 2).

Switching template replaces the cap geometry, the pile coordinates and the reinforcement; the loads and any linked column are kept. "Analysis only" (linked to the Reinforcement design switch in Materials) reports the pile reactions and EC7 pile checks without designing the cap reinforcement.

The cap design checks (EC2) are: bending at the column faces by beam theory; strut-and-tie (6.5) with strut, node and tie checks; anchorage of the bottom bars at the piles (9.8.1, 6.5.4, 8.4); beam shear at $\phi/5$ inside the pile faces with the 6.2.2(6) load reduction; punching at the column and at each pile (6.4); 45° corner-pile shear; and crack width (7.3.4). The pile positional tolerance (EC2 9.8.1(1)) is built into the lever arms, shear distances and punching reach.

The NHBC tree-depth check does not apply to pile caps (the NHBC tab is disabled for them).

3.4 Inputs - pad

Units are as shown on screen. Defaults are those of a new pad. All loads start at zero: a new pad is designed only for the loads you enter.

Geometry

Input	Unit	Default
Pad length (X)	m	3.0
Pad breadth (Y)	m	3.0
Pad thickness	m	1.87
Depth of soil over the pad	m	0.6
Column size	m	0.4×0.4
Column offset from pad centre	m	0, 0

The column size is set from the plan (right-click) and from a linked column node; plan dimensions can be edited directly on the plan.

Bearing Capacity card

Input	Unit	Default	Notes
Allowable bearing pressure (SBC)	kN/m²	250	A given value from the site investigation, not computed by the tool
Net bearing allowance	on/off	off	ON adds the overburden relief ($\gamma_{soil} \times$ depth of soil removed) to the SBC, so the SBC is read as a net value. Leave OFF unless the report gives a net allowable pressure
Action components	choice	Correlated G / Q / W	Or "Independent component envelope" (more conservative); see "Action basis"

A "guide" button beside the SBC opens a typical-values table (dense gravel above 600 kN/m² down to soft clays below 75 kN/m²). These are guidance values only.

Materials card

Input	Unit	Default	Notes
Soil friction angle φ'	degrees	30	Feeds the sliding check
Unit weight of soil	kN/m³	18	Weight of the soil over the base, and the net bearing allowance
Soil modulus E_s	MPa	20	Settlement estimate (and the stiffness test on a combined footing)
Water below GL	m	0	0 means no water table recorded

Input	Unit	Default	Notes
Interface friction δ/φ'	ratio	$\delta = 20°$ ($2/3$ of the default φ')	Choose 1.00 (cast against soil) or 0.67 (precast or smooth); EN 1997-1 6.5.3(10)
Unit weight of water	kN/m³	9.81	Choose 9.81, 10.00 or 10.30
Concrete grade		C30/37	The grade list is the carbon table's grades
Unit weight of concrete	kN/m³	25	Choose 24, 25 or 26
Reinforcement design	on/off	**off**	See "Plain concrete or reinforced"
Reinforcement grade		B500B	B500A, B500B, B500C (f_{yk} 500), B450C (450), B400 (400)
Top reinforcement	on/off	off	For hogging, uplift or lift-off
Design working life	years	50	50 or 100; EC2 Table 4.3N structural class
Special QC of concrete production	yes/no	no	EC2 Table 4.3N, one class lower
Exposure class (BS 8500-1)		XC2, wet, rarely dry	Sets $c_{min,dur}$ and the lowest concrete class allowed
Bottom cast against		blinding	Formwork, blinding/prepared ground (at least 40 mm) or directly on soil (at least 75 mm)
Sides cast against		formwork	Formwork or directly against soil (at least 75 mm)
Cover mode		auto	Auto sets the minimum nominal cover from BS 8500 plus EC2 4.4.1 (the cover boxes are locked and show the minimum). "User-defined" lets you type covers, which are then checked against the minimum
Covers (bottom, side, top)	mm	In auto mode the BS 8500 + EC2 4.4.1 minimum is shown and locked. The stored data default is 30 mm for a user-defined cover	

$\gamma_C = 1.50$ and $\gamma_S = 1.15$, $\alpha_{cc} = 0.85$ for bending and 1.00 for shear, wind $\psi_0 = 0.5$, $\xi = 0.925$ in 6.10b and the Design Approach 1 partial-factor sets are fixed and are not inputs. Everything that depends on circumstances (imposed-load category, punching β floor, δ/φ', γ_w, γ_c) is a drop-down with its range printed beneath.

Load cards

Three load groups, each with the same layout: horizontal X and Y and vertical Z in kN, and applied moments M_x and M_y at the column base in kNm.

Card	Meaning
Superimposed Dead Load (SDL)	Permanent unfactored actions from the column
Live Load (LL)	Variable imposed actions
Wind Load (WL)	Variable wind actions
Surcharge Loads	Permanent and variable surcharge on the ground over the base, kN/m²

Sign convention: positive Z is downward compression, negative Z is uplift. Horizontal signs follow the global X and Y axes.

The base's own self weight and the soil over it are computed by Civil ET and added to every combination. Do not include them in the SDL.

Each group can be switched in or out of every check with the "Use load" buttons (SDL, LL, WL) in the bar under the plan or section card. A switched-out group keeps its values but is ignored everywhere. All loads act at the column, wherever the column has been moved to (the old per-group "move with column" switches were removed).

When a load group is being driven by a live node link, its boxes are disabled with a note saying so. Delink the wire or switch off that transfer field to edit it by hand.

A snow load group exists in the data but is not an input on the pad, combined footing or strip. It is shown only if a snow component has actually been entered, and the pile cap panel still carries a snow field.

Punching Shear and Calculation Options card

Input	Default	Notes
Punching moment in expression (6.51)	Reduced by the soil reaction ($M_{Ed,red}$)	Alternative: full column moment, as EC2 is written
Minimum punching factor β	1.00	Options 1.00, 1.15 (internal), 1.40 (edge), 1.50 (corner). β is always calculated from the moments and floored automatically where a free edge cuts the perimeter; this only sets a higher floor
STR combination expression	6.10	Or 6.10a/6.10b with $\xi = 0.925$ (UK NA). Affects the structural design only (bending, shear, punching; the cap design on a pile cap). Geotechnical checks stay on 6.10

Reinforcement card (shown while reinforcement design is on)

Input	Unit	Default
Reinforcement type	bar or mesh	bar
Bottom bars X, Y: diameter / spacing	mm / mm	25 / 150
Top bars X, Y: diameter / spacing	mm / mm	12 / 150
Mesh (bottom, top)	BS 4483 mesh	A393, A252 (the table also holds A193, A142, A98, B1131 to B196, C785 to C283, D98, D49)
Crack-width limit w_{max}	mm	0.3
Quasi-permanent factor ψ_2		0.3 (set by the imposed-load category)
Aggregate size	mm	20 (options 10, 14, 20, 32, 40); sets the minimum clear bar spacing in the detailing check

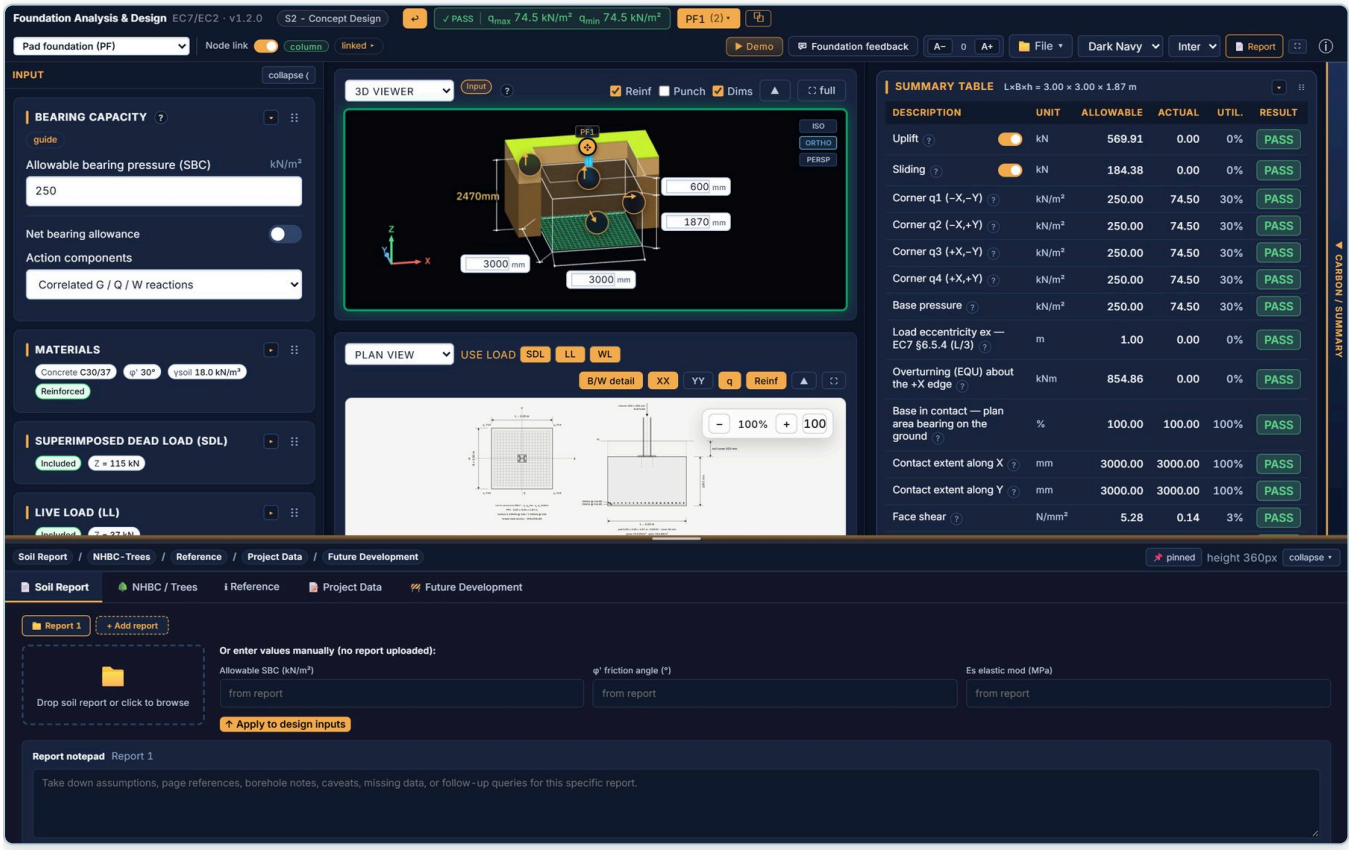
The outer bar layer is selectable, and bar counts are the whole bars that fit between the side covers. A bar size, spacing or f_{yk} that had to be substituted (a cleared box) is named on the panel and in the report and fails the design; it is not silently replaced.

Other inputs

- **Soil Report tab (bottom drawer).** Attach a ground investigation PDF and type the values it gives (SBC, φ', E, γ) into the "from report" boxes; they are written to the inputs when you apply them. The values are yours to enter; the tool does not interpret the PDF for you.
- **NHBC / Trees tab.** NHBC Chapter 4.2 tree-influence and minimum founding depth, from plasticity index (class or modified PI), tree species, distance and height, with the Table 4b/Table 5 minimum depths. The result becomes a

required founding depth.

- **Carbon panel (right drawer).** Cement type, concrete and steel EPD source, A4 transport and A5 construction/waste percentages (12 % and 4 % of A1-A3 by default; the A4 list also offers 3, 5, 8, 10, 15 and 20 %).
- **Project Data.** Project name, number, designer, checker, client, date and RIBA stage are entered once in "Project info & metadata" in the file menu and appear on every report.



Screenshot: The Soil Report tab in the bottom drawer of the Foundation module: attach a ground investigation report and enter the values it gives.

3.5 Calculation - what is checked

All figures below are for a pad foundation. The standards named are those the calculation uses.

Action basis

Two modes are offered on the Bearing Capacity card.

- **Correlated G / Q / W reactions (default).** Each of G, Q and W is one action: its vertical, horizontal and moment components carry the same factor and switch in or out together, because one event delivers all of them (EN 1997-1 2.4.2(9)P). The leading variable action is taken in full and accompanying actions at ψ_0 (EN 1990 expression 6.10). The base, soil and permanent surcharge are a separate permanent action.
- **Independent component envelope.** Every vertical, horizontal and moment component is enveloped on its own sign: an adverse component at full γ_Q, a favourable one left out, and no ψ_0 reduction. A supplementary and more conservative envelope. It is stated as a scope note in the report, not raised as a failure.

Design Approach 1

- Combination 1: $\gamma_G = 1.35$ (1.00 favourable), $\gamma_Q = 1.50$ (0 favourable).
- Combination 2: $\gamma_G = 1.00$, $\gamma_Q = 1.30$, $\gamma_{\varphi'} = 1.25$ (applied to $\tan\delta$ in the sliding check).

For the structural design each combination is split into sub-cases (variable actions leading in turn, present or absent). The structural flexure and shear are enveloped over all of them; Combination 1 normally governs.

Bearing (EC7 6.5.2)

- The check compares the pressure from the **characteristic** actions with the allowable bearing pressure (SBC, plus overburden relief in net mode). The utilisation is

$$\text{Utilisation} = \frac{q}{\text{SBC}\,[+\,\text{overburden}]}$$

- The base pressure is the planar pressure inside the kern (the middle-third rhombus $6e_x/L + 6e_y/B \le 1$). Outside the kern the base is partly lifted: the contact plane is solved exactly (pressure plane in equilibrium with the vertical load and both moments over the part still bearing), so the contact area, the contact extents, the section wedge and the lift-off line on the plan all describe the same boundary.
- A characteristic **envelope** is taken over the arrangements in which the imposed load and the variable surcharge are each present or absent (wind components stay together). The check answers to the worse of the combination as entered and that envelope.
- Corner pressures q_1 to q_4 are reported on the plan and in the summary (signed, EC7 6.5.2). Inside the kern the corner check requires each corner to be non-negative and no greater than the allowable pressure.
- Contact must be valid for every characteristic arrangement. Lift-off alone is not an EC7 failure if the solved contact is valid; a resultant outside the base is reported through overturning, the eccentricity limit and uplift.
- Net uplift (nothing bearing) is reported as zero pressure with the reason stated.

Important: the tool compares the pressure with your SBC. It does not compute a bearing resistance from c, φ' and γ. The allowable pressure and the settlement limits from the site investigation are listed in the report as "not calculated here (engineer to confirm)".

Sliding (EC7 6.5.3)

$$R_{H,d} = V'_d \tan \delta_d$$

This is checked for Combination 1 and Combination 2 separately; the governing one is reported.

- V'_d uses permanent vertical load as favourable (1.00) and a downward variable load as favourable and omitted ($\gamma_Q = 0$). Upward actions use the unfavourable factors.
- Horizontal action is the worst resultant over all directions (the exact maximum over the on/off patterns of the components).
- δ_d on Combination 2 has $\tan\delta$ divided by $\gamma_{\varphi'} = 1.25$.
- A horizontal action with no design vertical load gives a utilisation sentinel of 999 (no friction to mobilise); the node card caps displayed utilisations at 999 %.
- A traditional working-load factor of safety ($\text{FOS} \ge 1.5$) is also reported for comparison. It is **not** an EC7 verification and is not part of the pass/fail verdict.

Overturning (EQU) and eccentricity

- **EQU**: rigid-body rotation about each base edge using UK NA to EN 1990 Table NA.A1.2(A): destabilising $\gamma_{G,dst} = 1.10$ and $\gamma_{Q,dst} = 1.50$, stabilising $\gamma_{G,stb} = 0.90$ and $\gamma_{Q,stb} = 0$. Gross (not buoyant) weights are used and the pore-water thrust on the underside is a destabilising permanent action. The traditional $\text{FOS} \ge 2.0$ is shown for comparison.
- **Eccentricity (EC7 6.5.4(1)P)**: the worst design eccentricity of the sub-cases, against one third of the width. The kern limit $L/6$ is shown beside it as the lift-off threshold. A sub-case with no net vertical load fails this check because the base is not in contact.

Uplift (EC7 2.4.7.4, UK NA)

Stabilising: gross weight of base and soil, permanent surcharge and downward dead column load at $\gamma_{G,stb} = 0.90$. Destabilising: the pore-water thrust $u \times A$ and any permanent column uplift at 1.10, imposed and wind uplift at $\gamma_{Q,dst} = 1.50$. Downward variable loads are stabilising variable actions and take $\gamma_{Q,stb} = 0$. The working shows every term.

Groundwater

If a water level is entered, the pressures that drive the checks use effective (buoyant) weights below the water table. The gross values are kept for the quantity and carbon take-off and for the uplift check.

Settlement (EC7 Annex F.2)

Immediate (elastic) settlement of a rigid rectangular footing:

$$s = \frac{p\, b\, f}{E_m}$$

with the coefficient f interpolated in $\log(L/B)$ between tabulated values (1.0 gives 0.82, 1.5 gives 1.06, 2 gives 1.20, 3 gives 1.42, 5 gives 1.70, 10 gives 2.10), Poisson ratio 0.30 and E_m from the soil card. The result is compared with a **25 mm** indicator. It is labelled "Informational" and is **not** part of the overall pass/fail verdict. Settlement from the soil report can be recorded against the pad size it was quoted for.

Reinforced design (EC2)

With reinforcement design on, each of these runs and gates the verdict:

Check	Basis
Flexure	The base is analysed as a beam on its soil reaction in each direction; upward design pressure planar inside the kern and triangular outside; downward factored weight of base, soil and surcharge plus the factored column loads and moment at their positions. Sagging at the column faces, hogging wherever the diagram gives it. All DA1 sub-cases enveloped. Section design to EC2 6.1; $A_{s,min}$ and $A_{s,max}$ to 9.2.1.1
Top steel	Designed for hogging; otherwise the nominal minimum (9.2.1.1) where a top mat is asked for or the base lifts off. Where hogging appears and no top mat was asked for, the top face is first checked as plain concrete
One-way shear	EC2 6.2.2 at d from the column face, no shear reinforcement. An optional "transverse shear" basis ($\beta = a_v/2d$) is off by default because it is less conservative
Column-face shear	Against $v_{Rd,max}$, EC2 6.4.5(3)
Punching	EC2 6.4: control perimeters out to $2d$ against $v_{Rd,c} \times 2d/a$ with the net upward pressure inside each perimeter deducted (6.4.4(2)). Perimeters are truncated at the free edges, so a column near an edge is designed as an edge or corner support (β floor from Figure 6.21N). Face and perimeter can govern in different action cases; both traces are kept
Anchorage and detailing	EC2 8.4 anchorage with BS 8666:2020 scheduling; 9.8.2.2 for the bottom bars of a pad. Detailing rows check bar size (at least 8 mm), cover against bar diameter, clear spacing (the greater of bar diameter, aggregate size + 5 mm and 20 mm) and maximum pitch (the lesser of $2h$ and 250 mm)
Crack width	EC2 7.3.4, expressions (7.8) to (7.14), under the quasi-permanent combination $G + \psi_2 Q$ (wind $\psi_2 = 0$), against your w_{max}. Tables 7.2N and 7.3N are shown alongside

Plain concrete or reinforced

A new pad starts with **Reinforcement design off**. It is then designed as a plain (mass) concrete footing: EC2 12.9.3(1),

$$0.85\,\frac{h_F}{a} \geq \sqrt{\frac{3\,\sigma_{gd}}{f_{ctd,pl}}}$$

with

$$f_{ctd,pl} = \frac{\alpha_{ct,pl}\, f_{ctk,0.05}}{\gamma_C}$$

and $\alpha_{ct,pl} = 0.8$ (UK NA to 12.3.1(1)). σ_{gd} is taken as the largest DA1 design ground pressure. This criterion replaces the reinforced punching, one-way shear and crack checks; the column-face check is kept and every geotechnical check still applies. Bar detailing does not apply. Switch the option on in Materials to get the reinforced design.

Review issues ("code scope")

Civil ET raises a scope issue (and the pad then cannot pass) if, for example: the concrete class is outside C12/15 to C50/60; the column lies outside the footing; interface friction is outside 0 to φ'; f_{yk} is outside 400 to 600 MPa; the singly reinforced section limit is exceeded; legacy moments were entered without a load-case source. An elongated column (aspect ratio over 2) raises a review note about wall-like shear. These appear in the results panel and in the report.

3.6 Outputs

The summary table at the top of the calculation column lists each check with the design value, the resistance, the utilisation and PASS/FAIL. Each row has a "?" or "where" link to the card that holds its working.

Output	Where
Overall verdict	Header, summary, report cover, node card colour
q_{max}, q_{min}, q_1 to q_4, contact area and extents, lift-off line	Bearing Pressure card, plan view
Bearing, sliding, EQU, uplift, eccentricity utilisations	Sliding, Overturning and Uplift card
Punching and shear utilisations (face and perimeter)	Punching card
Required and provided steel, top steel, crack width	Bending Reinforcement and Top Reinforcement cards
Bar schedule (mark, shape, diameter, length, number; BS 8666 rounding)	Detail drawing and report
Settlement	Settlement card (informational)
NHBC required depth and pass	NHBC tab and section view
Embodied carbon A1 to A5, steel and concrete quantities	Carbon panel and report
Alternative designs	Alternative designs card

The overall verdict is the AND of the code-scope, analysis, layout/detailing, bearing, corner, contact, eccentricity, sliding, uplift, EQU, face shear, punching (or plain-concrete) check, reinforcement, crack and one-way shear checks (the last two only when reinforcement is designed), the NHBC depth if one is set, and "no substituted inputs".

Alternative designs

The card "Alternative designs - reinforced concrete (or mass concrete), lowest carbon first" offers cards for other plan sizes, depths and (for a reinforced base) bar arrangements. Each card is a full design run and passes every check. They are ranked on embodied carbon. Clicking a card applies its size, depth and bars in one update.

- The search runs **only on the button** ("Find reinforced alternatives" or "Find mass concrete alternatives", then "Search again from the current design"). It never runs on an input change.
- Results stay valid while only plan size, depth, bars or column positions change. Any other input clears them.
- It runs in the background; a combined footing search can take 5 to 15 seconds.
- With reinforcement design off, the alternatives are mass concrete pads to EC2 12.9.3.

Views and the detail drawing

- **Plan view**: base pressure heat map with corner values, lift-off line, loads, reinforcement, punching zone, trees and eccentricity resultant (toggles on the toolbar).
- **Section view**: X-X or Y-Y cut with bearing pressure wedge, levels, NHBC data, and optional shear and moment sketches.
- **3D viewer**: the foundation, column, piles, reinforcement and trees.
- **B/W detail**: a black-and-white drawing sheet with the plan and section X-X or Y-Y, dimensions, bar callouts (marks B1/B2 bottom, T1/T2 top), corner pressures (toggle "q"), reinforcement (toggle "Reinf"), pile table for pile caps and a bar bending schedule band. The drawing works from the resolved bar layout, so the drawn bars always match the calculation, and it draws the bar count the design uses without capping it.

3.7 Multiple foundations and the Foundation node

The designer holds a list of foundations. The Foundation node on the node map is a view of that list.

- One input port per design. Pad and pile cap designs can receive data; combined and strip designs only keep output ports for now (the card says "Incoming data is enabled only for pad foundations and pile caps").
- The card shows each design's type and worst utilisation. The utilisation gathers every gating check (bearing, sliding, punching, face shear, EQU, eccentricity, uplift ratio, crack width, one-way shear, and lift-off), so the chip cannot read comfortably when the module reports FAIL. The card border takes the colour of the design currently selected.
- Node summaries show the designer's published result only.

Column to Foundation wiring

Draw a wire from a Column node's OUT socket to a Foundation input point (one pad or pile cap, or the main "all eligible" input). The column's design loads, size, material, section, grade and base plate size are written to the foundation. Loads and Math nodes (and IFC elements) can also feed pad and pile cap targets. A Base plate node can be linked both ways.

- Each foundation has a **"Use column node"** toggle in the header. Turn it off to stop that foundation receiving transfers.
- While a transfer drives a load group, those boxes are locked. The foundation remembers the manual loads so you can return to them.
- Combined and strip footings stay manual for column inputs.

The no-redesign-unless-numbers-change rule

A finished design is never recalculated just because a card was added, the map was zoomed, or the design was opened again; only a change to its numbers redesigns it.

Practical effect: moving around the map and re-syncing a wire is cheap. If a design shows "Updating design..." it is because its numbers changed.

3.8 Reports

The Report button (header) opens the foundation report window.

- **Report mode**: "Detailed report" (full working) or "Simple report" (checks, figures, key inputs and carbon totals).
- **Colour scheme**: Amber engineering, Black / white, Corporate blue, Reading comfort.
- **Calc description / scope**: free text that appears in the report header (applied after a short pause).
- A live preview of the text note, and a gallery of the figures exactly as they will be exported.
- Buttons: Regenerate, Copy, .txt, Word (.doc), PDF (opens the print dialogue; choose Save as PDF).

Detailed report chapters: Project Details; Summary of Checks; Foundation Views and Design Diagrams; Input Data; Working Calculations; Sustainability and Embodied Carbon Assessment; References and Design Basis. The simple report has Project Details, Summary of Checks, Foundation Views, Key Inputs, Carbon Summary, and References.

Figures in the foundation report: the 3D view, the B/W detail drawing (X-X with plan, and Y-Y in the detailed report), the plan, sections X-X and Y-Y (detailed), the real shear-force and bending-moment diagrams where bars are designed, and the NHBC graphic. Reports follow the actual design: reinforcement content is printed only when reinforcement design is on (a pad with design off is reported as plain concrete to EC2 12.9.3; an unreinforced strip as plain concrete; a pile cap in analysis-only mode has no cap design) and the version number is 1.2.0. The full rules and the other modules' reports are in the Reports chapter.

3.9 Tips and known caveats

- The tool compares pressure with a **given SBC**. It never derives a bearing resistance from c, φ' and γ. Settlement is an indicator against 25 mm only.

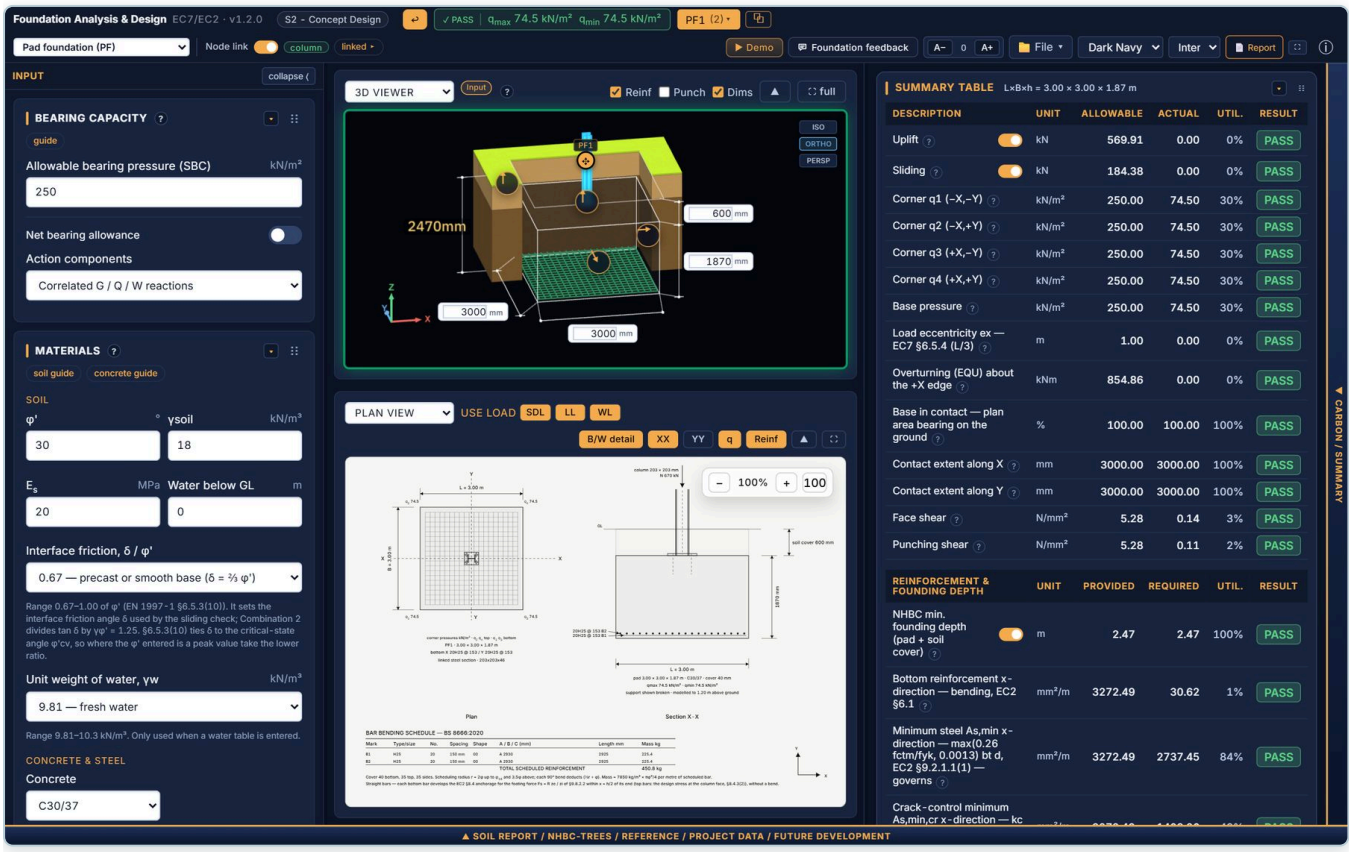
- New pads are plain concrete until you switch Reinforcement design on. If a pad you expected to be reinforced shows a thickness check "$0.85\,h_F/a$", that is why.
- Cover defaults to the BS 8500 minimum for the chosen exposure class ("auto"). Switch the cover mode to user-defined to type your own; they are then checked against the minimum and a shortfall is flagged "review", not "fail".
- Raft foundations are locked in the selector. A raft design is not offered in this release.
- Elongated columns (aspect ratio over 2) trigger a review note: the truncated perimeter may overstate resisting length.
- The 3-pile cap triangular option uses strut-and-tie; the others use beam theory with strut-and-tie reported alongside.
- The NHBC check needs a founding depth; the pad passes the check when $h + D_f$ is at least the required depth. It is disabled for pile caps.
- The checks assume what the report states under "Not calculated here": allowable bearing and settlement limits from the SI, design groundwater, ground chemistry (BS 8500 DC class / ACEC), restraint cracking and the column starter / load transfer detail.
- Undo: a global Ctrl+Z / Ctrl+Y manager covers the foundation (history per design); text boxes keep their native undo.
- Touch and pen are supported.

> **References.** The standards, guidance and data sources used in this chapter are listed, with web links, in Chapter 17.

3.10 Screen reference - panel by panel

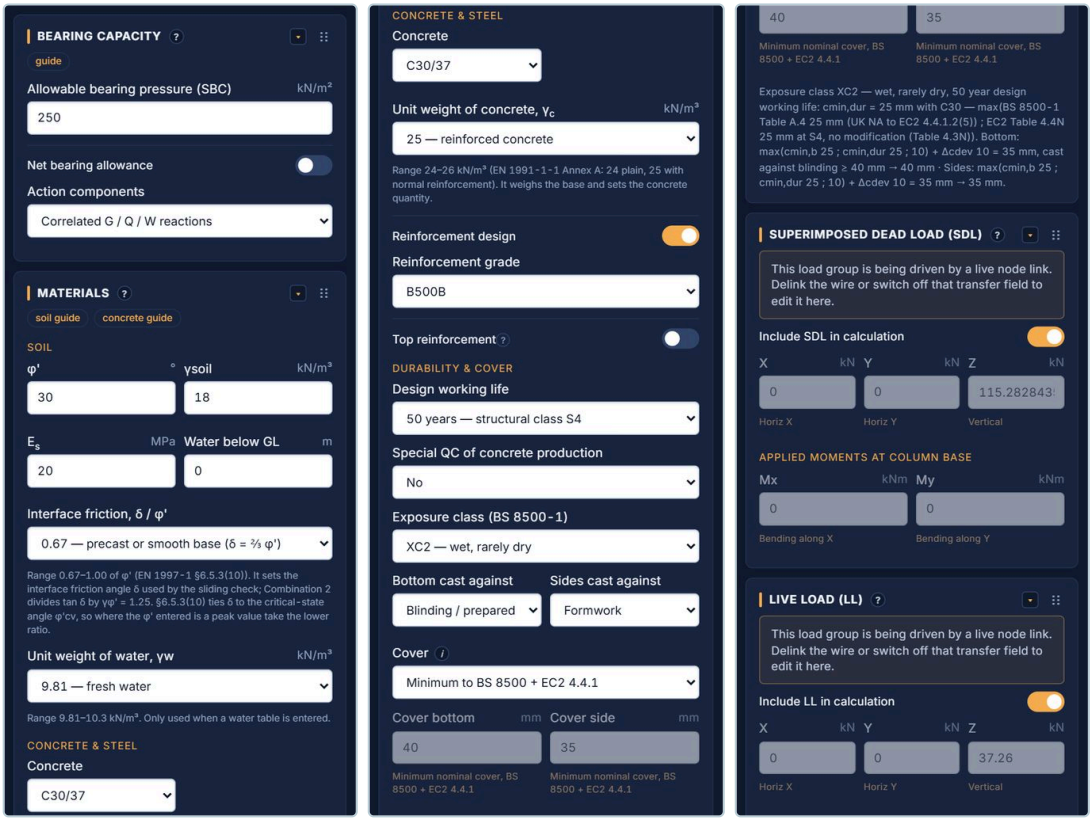
These screenshots show every panel of the module exactly as it appears on screen, scrolled from top to bottom, so you can match any field, button or result to the explanations in this chapter. Cards that are collapsed by default are shown expanded.

Pad foundation (PF): the whole screen

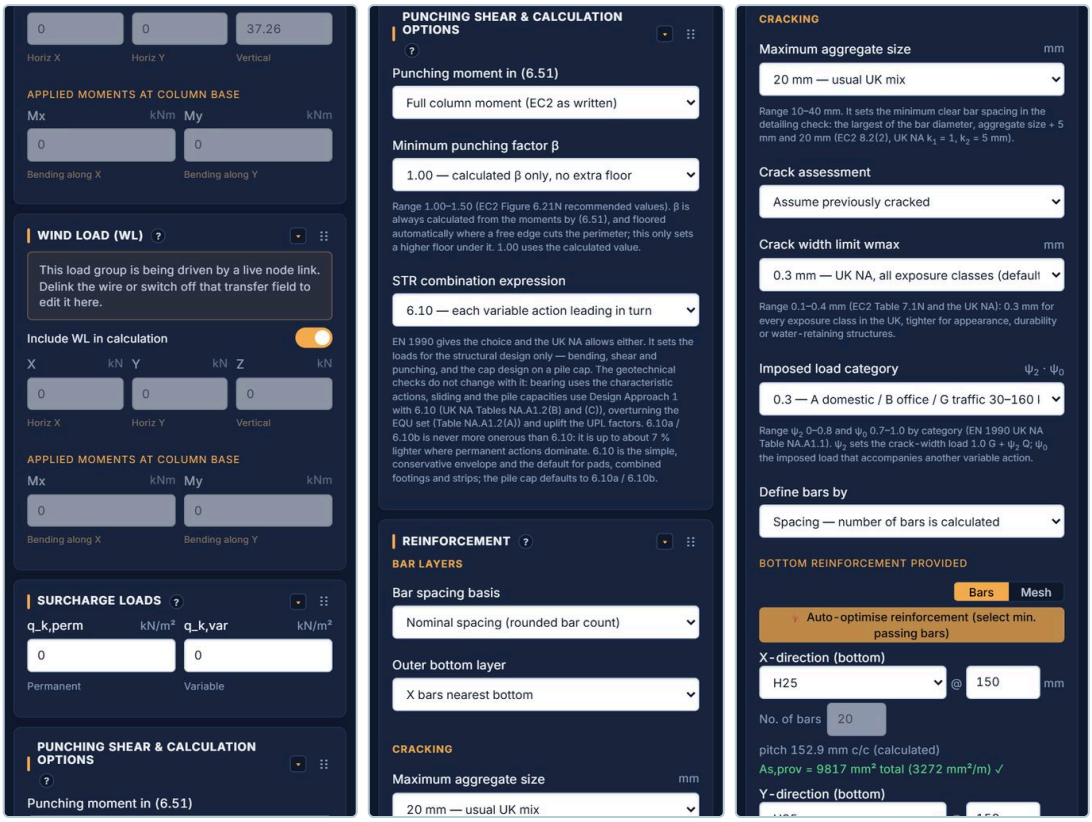


Pad foundation (PF): the whole screen.

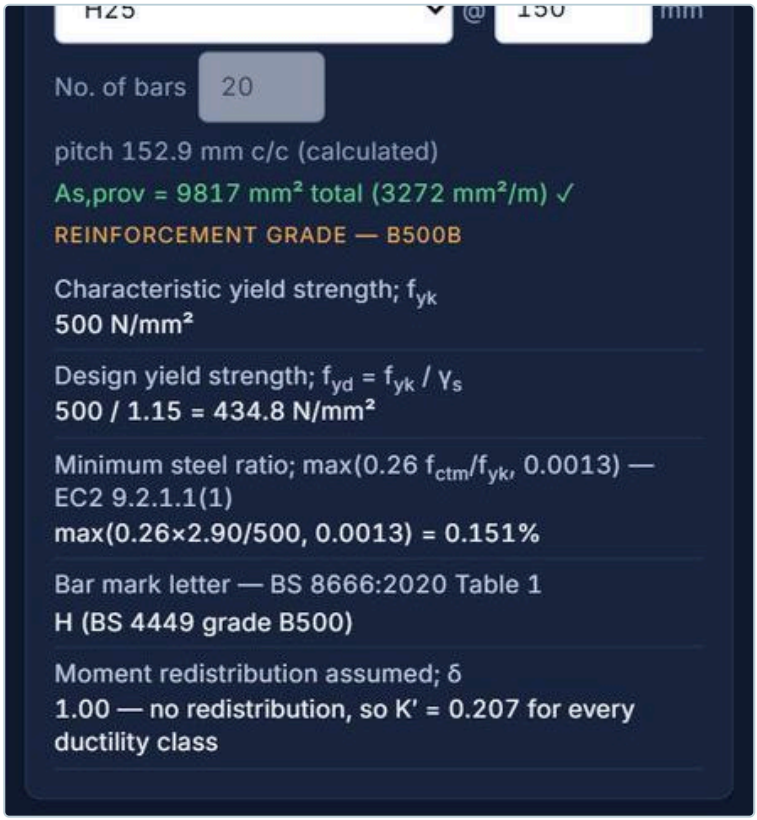
Pad foundation (PF): input panel (left of the screen)



Pad foundation (PF), input panel (left of the screen), parts 1-3 of 7. Cards shown: Bearing capacity, Materials, Superimposed dead load (SDL), Live load (LL).



Pad foundation (PF), input panel (left of the screen), parts 4-6 of 7. Cards shown: Wind load (WL), Surcharge loads, Punching shear & calculation options, Reinforcement, Bar layers, Cracking.



Pad foundation (PF), input panel (left of the screen), part 7 of 7.

Pad foundation (PF): results panel (right of the screen)



Pad foundation (PF), results panel (right of the screen), parts 1-2 of 6. Cards shown: Bearing pressure — EC7.

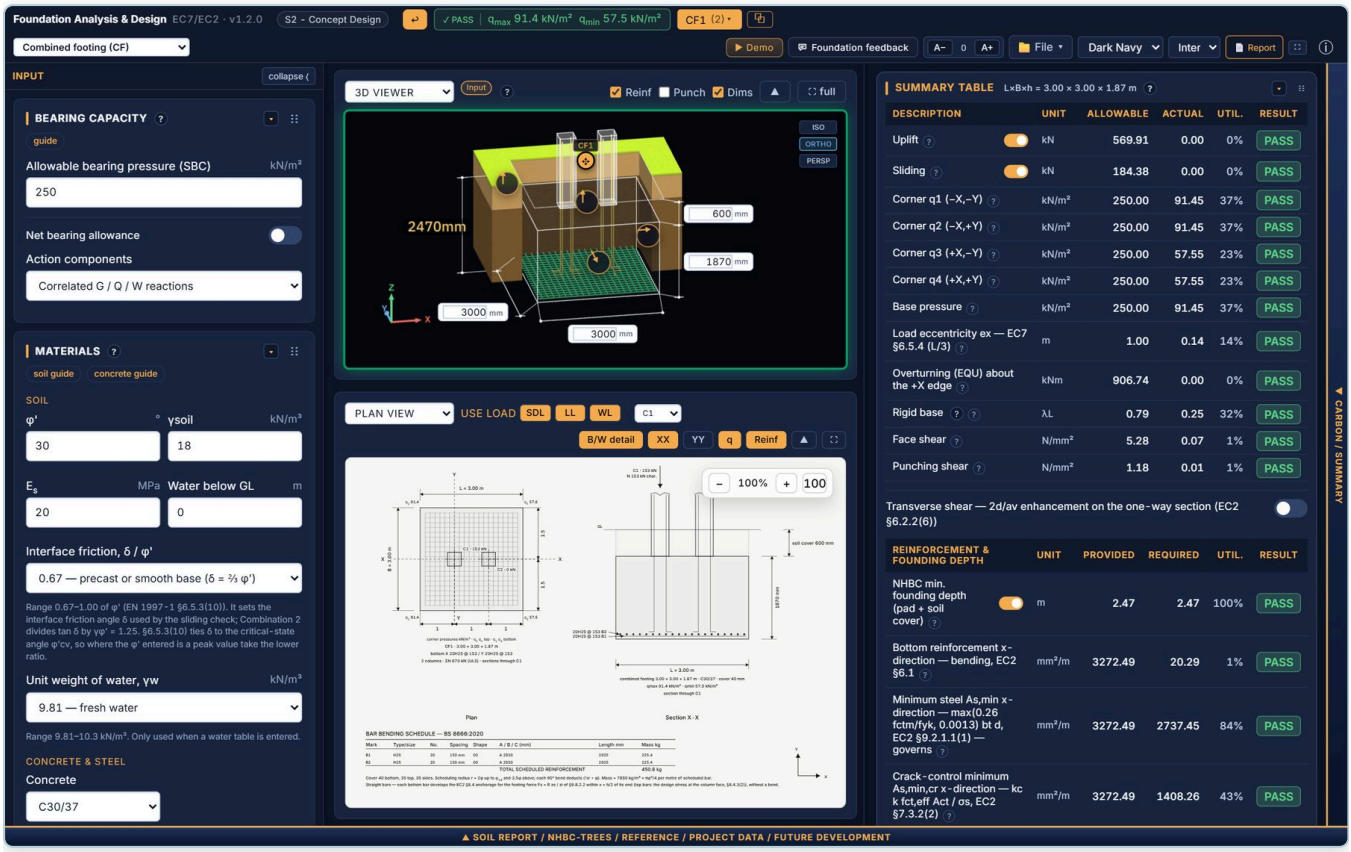


Pad foundation (PF), results panel (right of the screen), parts 3-4 of 6. Cards shown: Bearing pressure — EC7, Sliding, overturning & uplift — EC7.



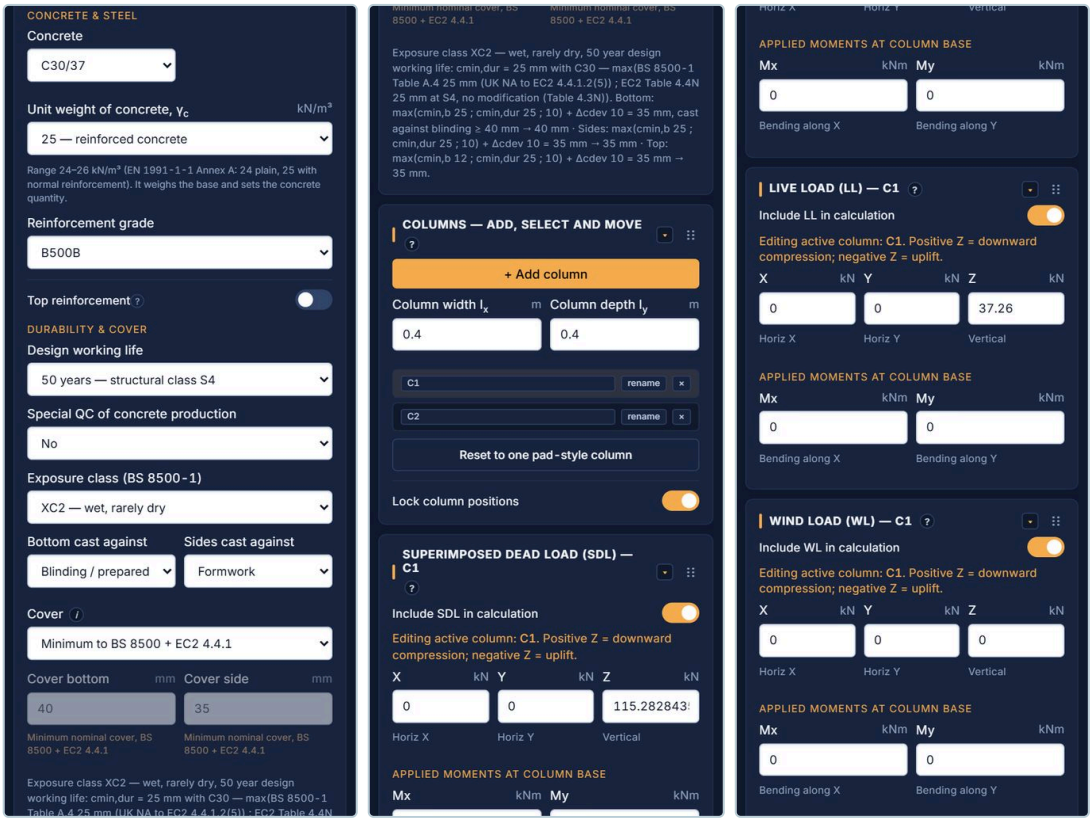
Pad foundation (PF), results panel (right of the screen), parts 5-6 of 6. Cards shown: Bending reinforcement — EC2 (bottom), Settlement — informational.

Combined footing (CF): the whole screen

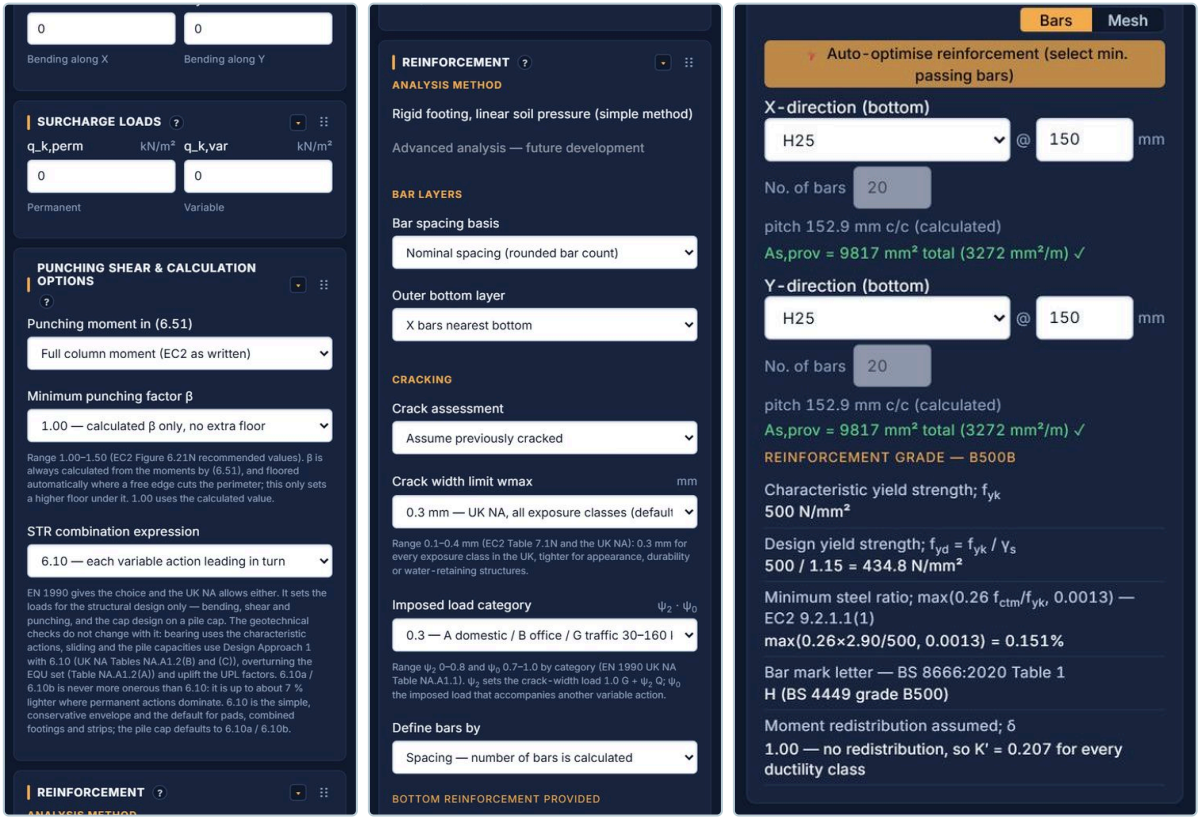


Combined footing (CF): the whole screen.

Combined footing (CF): input panel (left of the screen)



Combined footing (CF), input panel (left of the screen), parts 1-3 of 6. Cards shown: Columns — add, select and move, Superimposed dead load (SDL) — C1, Live load (LL) — C1, Wind load (WL) — C1.



Combined footing (CF), input panel (left of the screen), parts 4-6 of 6. Cards shown: Surcharge loads, Punching shear & calculation options, Reinforcement, Analysis method, Bar layers, Cracking.

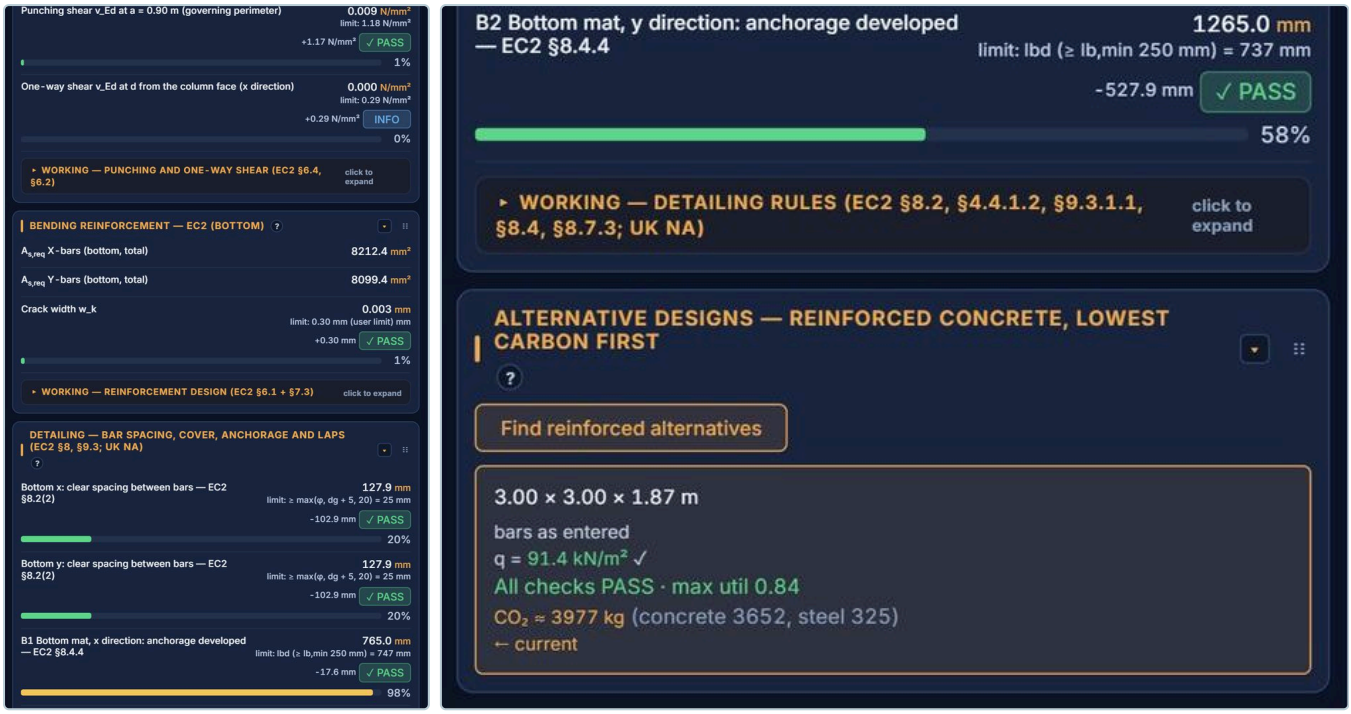
Combined footing (CF): results panel (right of the screen)



Combined footing (CF), results panel (right of the screen), parts 1-2 of 6. Cards shown: Bearing pressure — EC7.

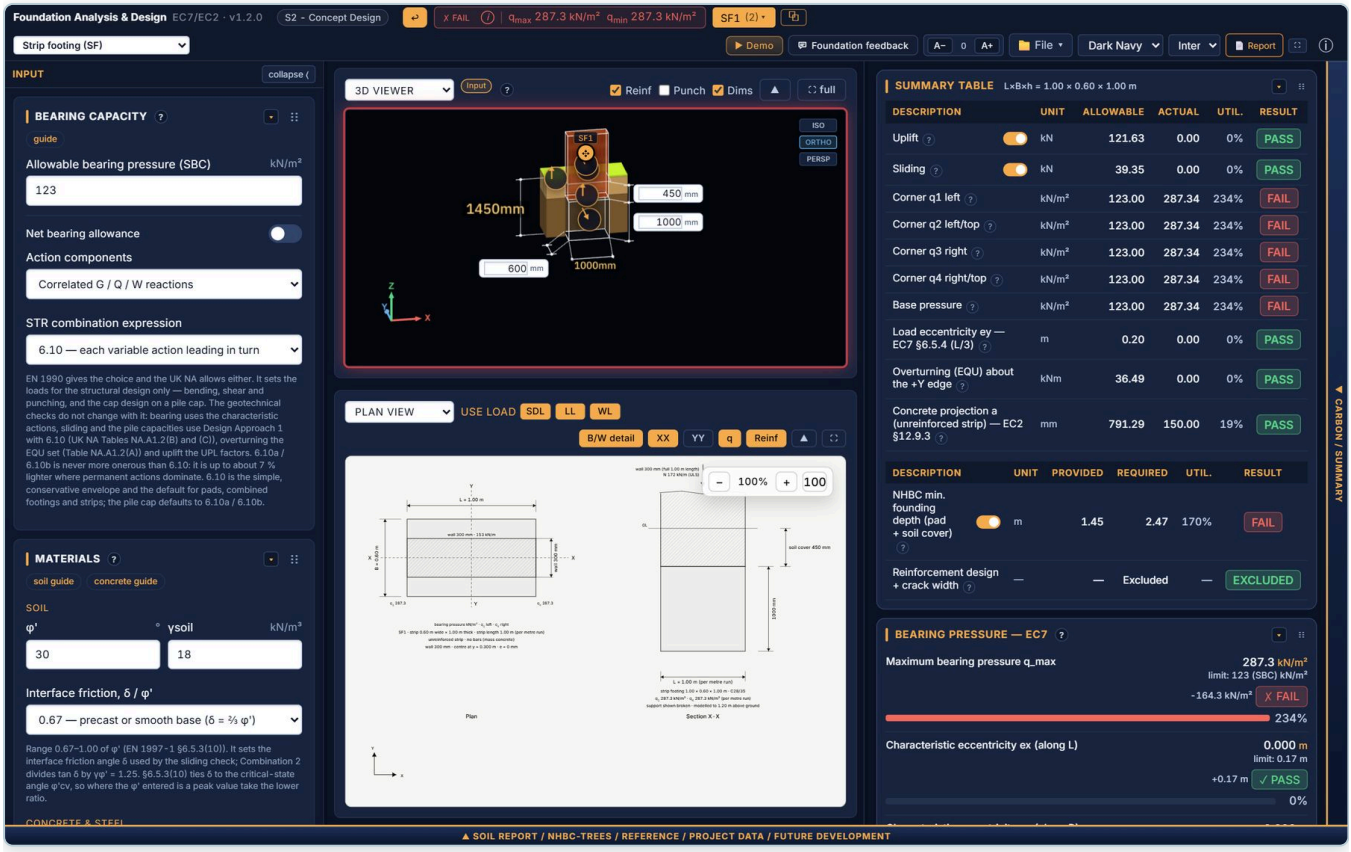


Combined footing (CF), results panel (right of the screen), parts 3-4 of 6. Cards shown: Sliding, overturning & uplift — EC7.



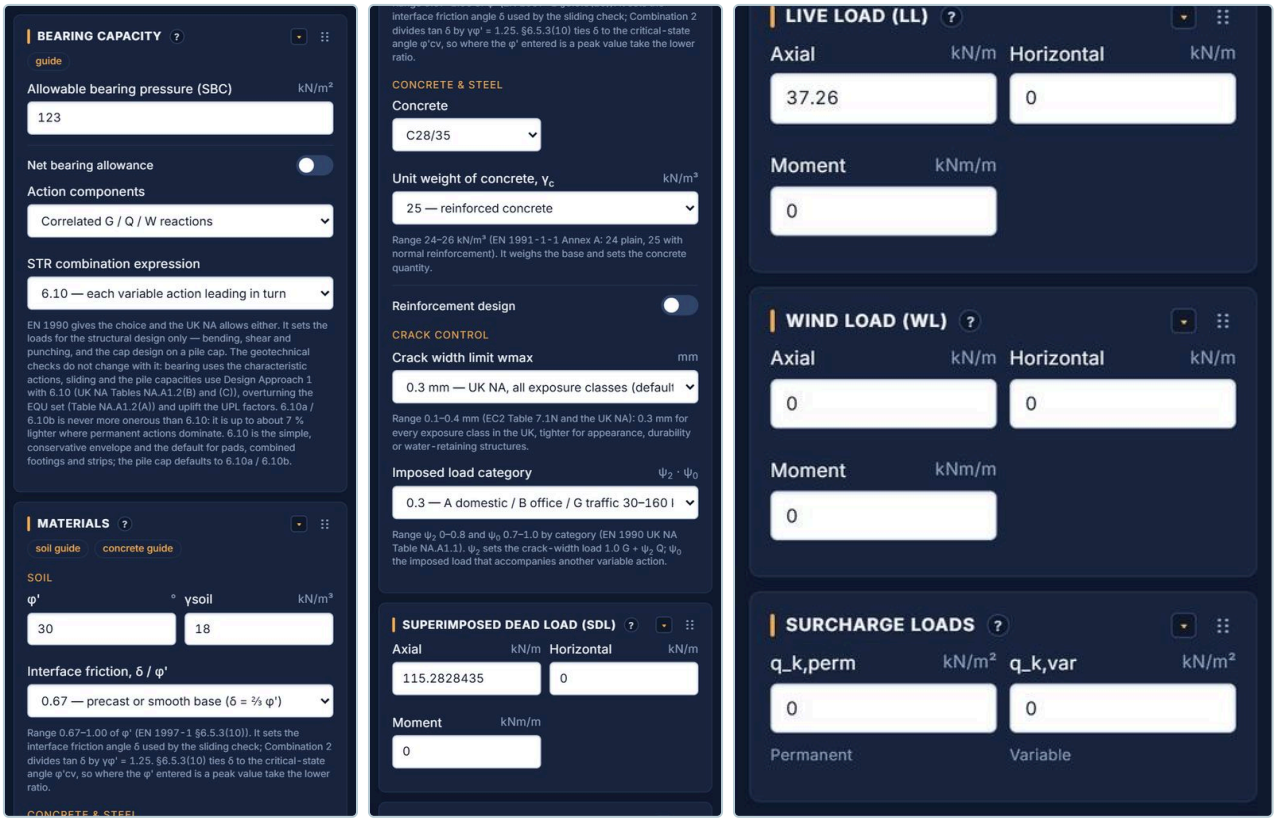
Combined footing (CF), results panel (right of the screen), parts 5-6 of 6. Cards shown: Bending reinforcement — EC2 (bottom).

Strip footing (SF): the whole screen



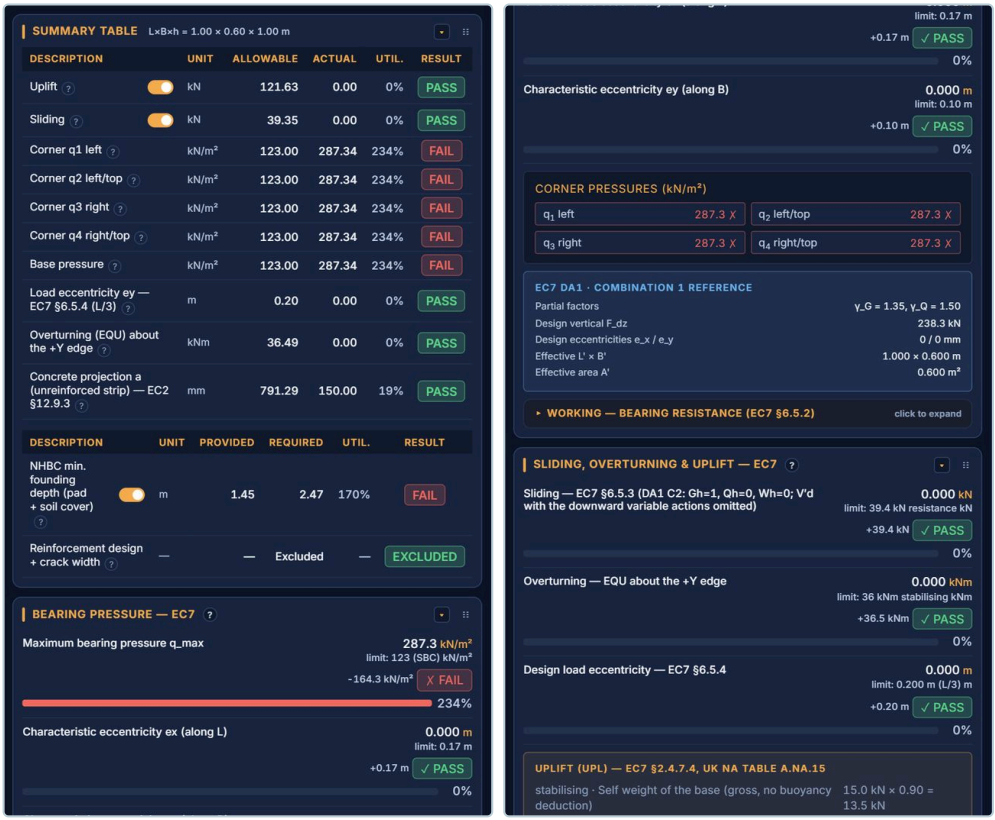
Strip footing (SF): the whole screen.

Strip footing (SF): input panel (left of the screen)

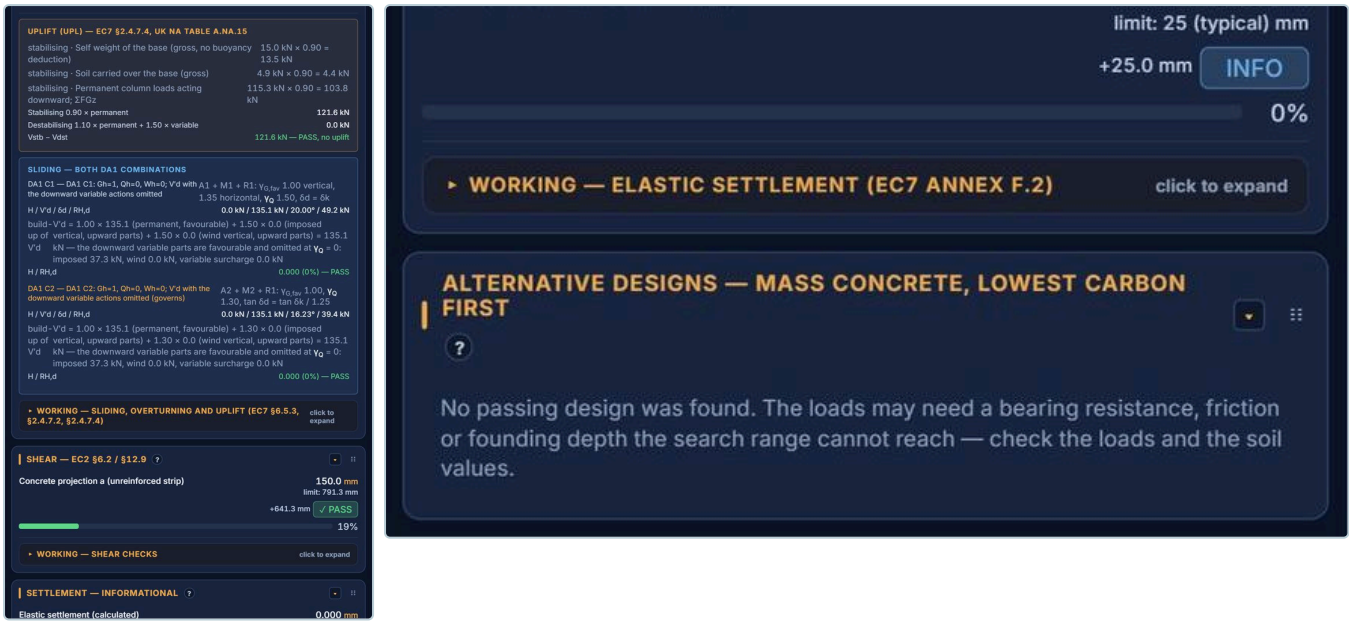


Strip footing (SF), input panel (left of the screen), parts 1-3 of 3. Cards shown: Bearing capacity, Materials, Superimposed dead load (SDL), Live load (LL), Wind load (WL), Surcharge loads.

Strip footing (SF): results panel (right of the screen)

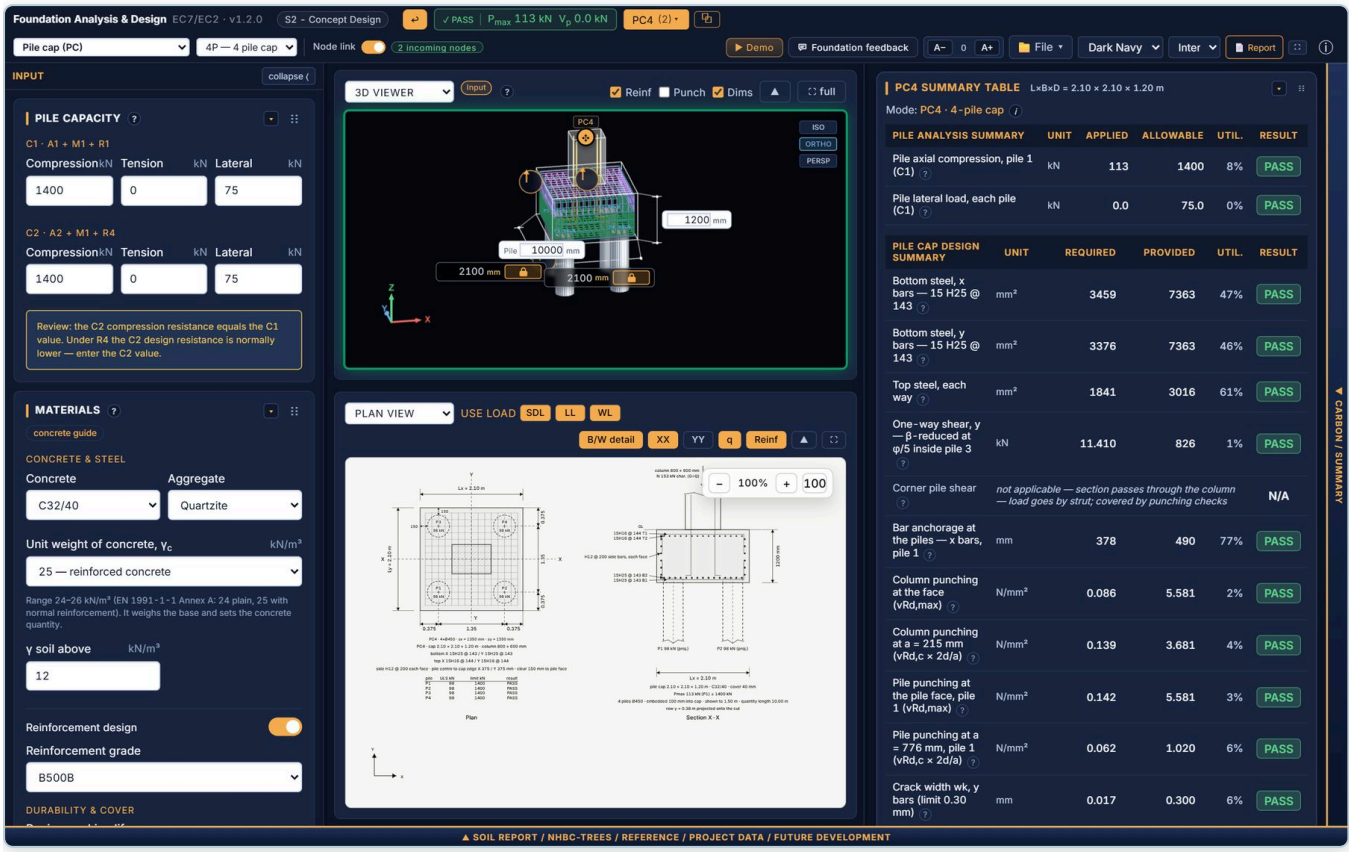


Strip footing (SF), results panel (right of the screen), parts 1-2 of 4. Cards shown: Bearing pressure — EC7, Sliding, overturning & uplift — EC7.



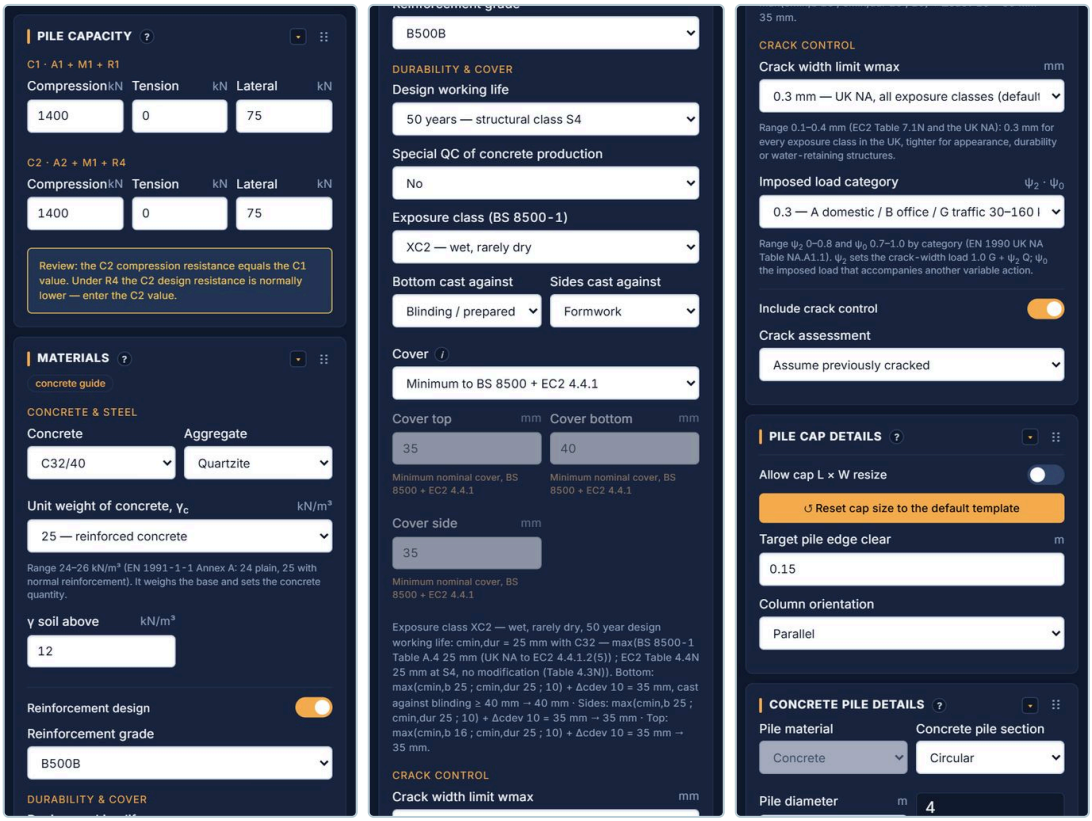
Strip footing (SF), results panel (right of the screen), parts 3-4 of 4. Cards shown: Settlement — informational, Alternative designs — mass concrete, lowest carbon first.

Pile cap (PC): the whole screen

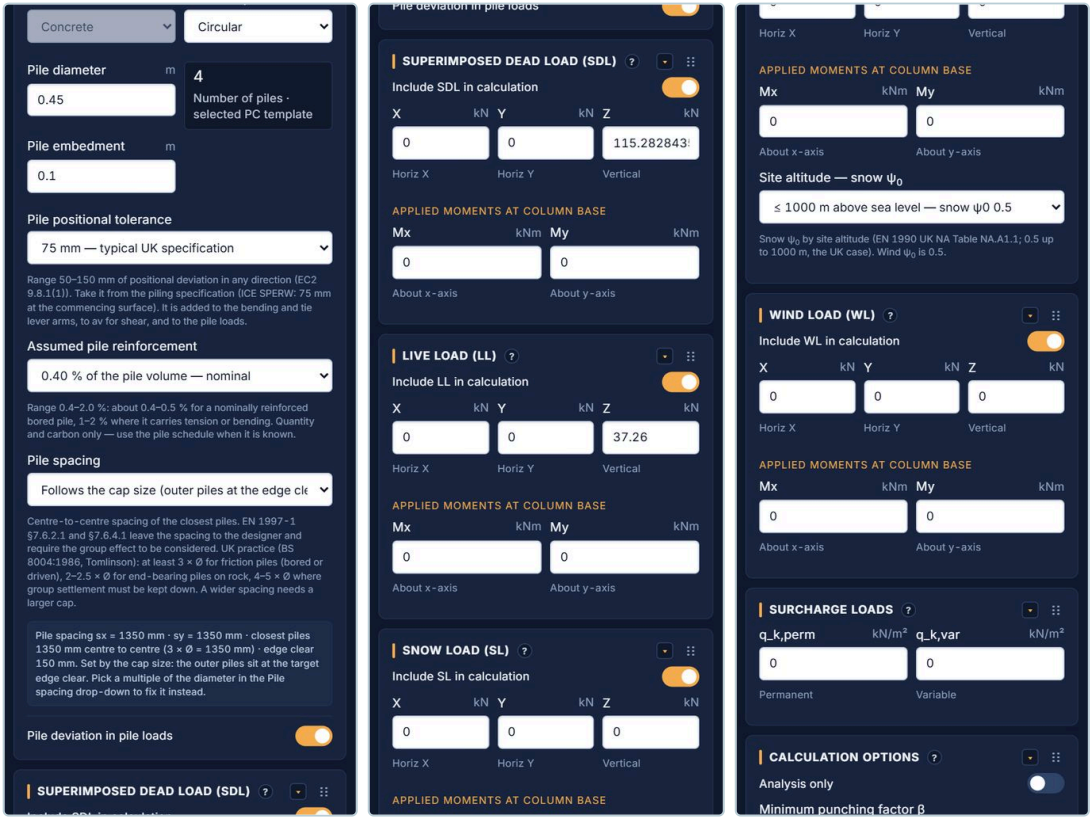


Pile cap (PC): the whole screen.

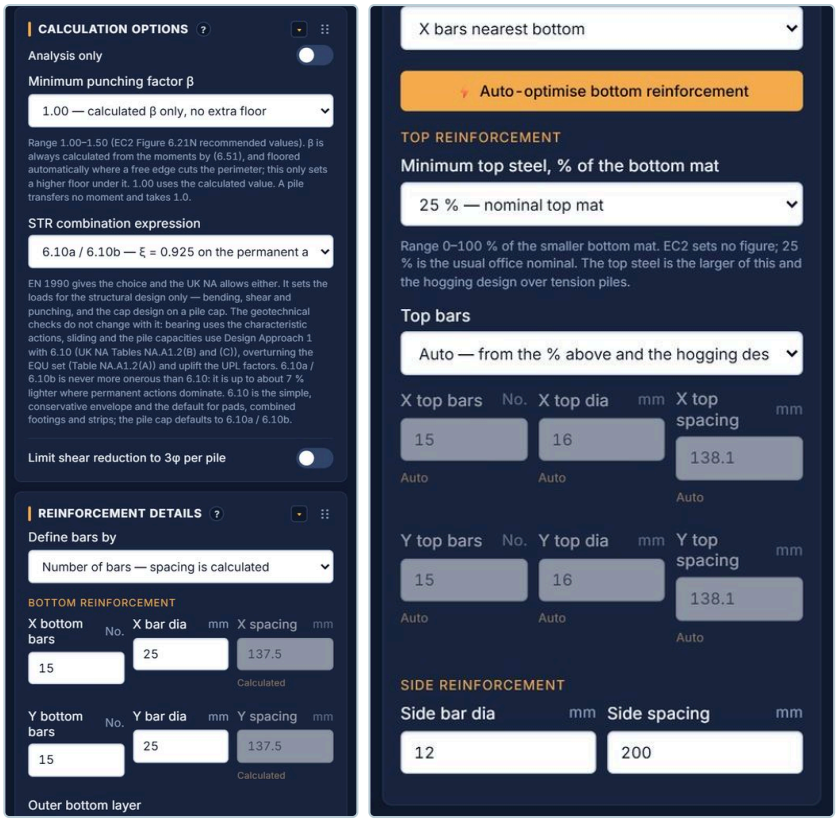
Pile cap (PC): input panel (left of the screen)



Pile cap (PC), input panel (left of the screen), parts 1-3 of 8. Cards shown: Pile capacity, Materials, Pile cap details, Concrete pile details.



Pile cap (PC), input panel (left of the screen), parts 4-6 of 8. Cards shown: Superimposed dead load (SDL), Live load (LL), Snow load (sl), Wind load (WL), Surcharge loads, Calculation options.



Pile cap (PC), input panel (left of the screen), parts 7-8 of 8. Cards shown: Calculation options, Reinforcement details, Bottom reinforcement, Top reinforcement, Side reinforcement.

Pile cap (PC): results panel (right of the screen)



Pile cap (PC), results panel (right of the screen), parts 1-2 of 3. Cards shown: Pile analysis — EC7 reactions, Pile cap design — EC2.



Pile cap (PC), results panel (right of the screen), part 3 of 3.

CHAPTER 4

Column design

The Column node designs a single column, in either structural steel (EN 1993-1-1) or rectangular reinforced concrete (EN 1992-1-1). You give it the loads (by case or as one set of design actions), the section and the effective lengths. It returns a pass/fail utilisation for every check, a full set of working calculations, live section views, embodied carbon, and a Word/PDF/text report. It can take its loads from beam, slab, IFC and Sketch-ET nodes, and it passes its loads, size and utilisation on to the Foundation and Base plate nodes.

This chapter describes what the node does as built in v1.2.0. Clause numbers quoted are those used in the calculations and reports.

4.1 Opening and adding a column

- Column is one of the three core nodes (Foundation, Column, Base plate) that exist on a new node map. More can be added from the node library (tag "EC3 / EC2").
- Open the node from the map to get the full Column workspace: an input panel on the left (collapsible, resizable), and the results, views and working calculations in the main area.
- One Column node can hold several column designs (C1, C2, ...). Each design is a row on the node card with its section and utilisation, and each row has its own input and output dots, so different wires can feed different columns. A header picker switches the active design.
- Each design has its own state (material, section, loads, geometry). Designs are stored in the browser (local storage), together with the rest of the workspace.

A design can be steel or concrete. The first control in "Column details" is **Design material** (steel / concrete). Everything else in the panel changes with it.

4.2 Load model (load take-down)

4.2.1 Two ways to enter loads

The **Design loading** card has a switch, "Enter loads by case (DL / LL / WL)":

Mode	What you type	What the checks use
ON (default)	A grid of dead (DL), imposed (LL) and wind (WL) rows for each action: P/Z (axial, vertical), V_x, V_y, M_x, M_y	The rows combined as described in 4.2.2
OFF	Single values N_{Ed}, V_{Ed} (y-y), V_{Ed} (z-z), and bottom moments about y-y and z-z	Exactly as typed. No partial factors are added, so enter factored ULS values

Axes: P/Z is vertical. V_x and M_x work about the section's y-y axis (bending on the depth h); V_y and M_y about z-z (on the width b). Compression is positive, tension negative.

Top moments. The grid's M_x and M_y rows are the moments at the **bottom** of the column. Type the moments at the **top** in the fields "Bending moment top - about y-y / z-z (kNm)". The larger of the two ends is treated as the end moment M_{02} (RC) or the larger end moment for the Annex B / equivalent-moment factors (steel); the end-moment ratio comes from the pair. Moments must be unredistributed elastic values.

4.2.2 Characteristic or design rows

When the grid is on, a selector "The row values are" chooses:

- **already factored - final ULS design actions, used as typed** (default). DL + LL + WL are simply summed. Any γ factors must already be inside your numbers.

- **characteristic (unfactored)**. Civil ET applies the EN 1990 Exp. (6.10) factors itself, and designs the column for every consistent load set (see below).

Wires from beams, slabs, IFC and Sketch-ET carry characteristic values, so connecting one switches the column to "characteristic" automatically and locks the selector.

In characteristic mode Civil ET builds these sets (G = DL rows, Q = LL rows, W = WL rows; UK NA Table NA.A1.2(B)):

Set	γ_G	Q factor	W factor
1	1.35	1.5	w_{Acc}
2	1.35	q_{Acc}	1.5
3	1.35	1.5	0
4	1.35	0	1.5
5	1.35	0	0
6 to 10	1.0	as sets 1 to 5	as sets 1 to 5

where $q_{Acc} = 1.5\,\psi_0$ of the imposed-load category and $w_{Acc} = 0.75$ (that is 1.5×0.5 for wind). The $1.0G$ sets matter where wind relieves the axial load (RC section below the balance point, or a column going into tension). Wind is taken in both directions (opposite sign also tried) unless the "wind reversible" setting is off. Duplicate sets are removed. The column is checked for every set and the **governing set** (highest utilisation, fails ranked first) is the one reported; the combination label (for example "$1.35G + 1.5Q + 0.75W$") is shown in the summary and report.

Imposed load category. In characteristic mode a selector sets the EN 1990 category (ψ factors from the beam node's table, UK NA Table NA.A1.1). ψ_0 is 1.0 for category E (storage) and 0.7 for the others; for category H (roof) imposed load and wind are not combined (EN 1991-1-1 3.3.2(1)). Default category is B.

4.2.3 What feeds the rows (wires)

From	Valid	What arrives
Beam analysis	yes	End reactions (characteristic G and Q, and the shear/moment rows the beam wire is set to send). Several beams on one column are **summed**, as end reactions on one column add up
Slab design	yes	Characteristic area loads × a quarter-panel tributary area ($l_x/2 \times l_y/2$) as DL and LL axial. Adjust the grid if the real tributary differs
IFC viewer	yes	Axial DL and LL from the design tab (area load × tributary area × floors), plus the member's section and geometry
Sketch-ET (prelim)	yes	DL and LL column base reaction: a chosen sketched column, or the sum of all
Steel sections (Blue Book)	yes	The section only (family and section), see 4.3.1
Wind designer	the wire can be drawn	See the note below
Loads, Dimensions, Math	no	Not accepted by a column

Notes:

- A live wire locks the rows it writes (shown with a lock) and the "enter loads by case" switch is forced ON. Rows that the wire does not write stay yours, but they are then read as **characteristic** and factored together with the wired rows: re-check anything you typed before connecting.
- If a wire stops driving a row (for example you change what the beam sends) that row is set back to zero, so a stale load cannot linger.

- **Type my own loads instead** takes the load rows off a wire without disconnecting it. The wire still delivers geometry and section. **Use the model's loads again** hands control back. An unplugged (paused) wire no longer owns the loads.
- The map permits a wind-to-column wire, but the column does not read wind rows from it. Enter the wind rows by hand.

4.2.4 Self-weight

The column's own weight is **not** added to its own N_{Ed}. It is added when the column passes its loads on: to the foundation (added to the DL axial, see 4.9) and to the base plate (see Chapter 5). The weight is steel mass $\times$ length $\times$ g, or for RC the concrete and rebar mass the take-off produced.

4.3 Steel columns (EN 1993-1-1)

4.3.1 Section and material

Input	Default	Notes
Family	Universal columns (UC)	UB, UC, UBP, PFC, equal angles (L), CHS, SHS, RHS, EHS and others from the embedded Blue Book data
Section	203x203x46	Local Blue Book data is embedded in the node. A connected Steel sections node overrides family and section (a note says so)
Steel grade	S355	Yield and ultimate by thickness band (EN 1993-1-1 Table 3.1). "User defined" accepts f_y and f_u (defaults 355 / 470 N/mm^2)
Hollow forming	hot finished	Or cold formed (buckling curve c)
I fabrication	rolled	Or welded (different Table 6.2 and LTB curves)
γ_{M0}, γ_{M1}	1.0, 1.0	UK NA values; change only for another National Annex

An **Alternative steel sections - carbon optimised** card searches UB/UC/UBP/PFC/L/CHS/SHS/RHS/EHS for lighter or lower-carbon sections that pass, and can apply one. The search runs only when you press its button; results stay while the inputs are unchanged.

4.3.2 Length and buckling inputs

Input	Default	Meaning
System length (mm)	3500	One physical length; Y and Z fields mirror each other
Buckling mode, each axis	factor	`factor`: $L_{cr} = K \times L$. `frame`: K from the frame joints (below). `direct`: type L_{cr}
K factors / end restraint	1.0	End condition presets: both ends fixed 0.5, one end fixed 0.7, restrained 0.85, pinned 1.0, cantilever 2.0
Sway (major / minor)	off	Sway buckling mode: sets the Annex B equivalent moment factor C_m to 0.9 and uses the sway frame formula
K_T	1.0	Torsional buckling length factor (cl. 6.3.1.4). Hollow sections exempt
LTB moment shape	auto	auto: k_c from the end moments (Table 6.6), $C_1 = 1/k_c^2$. uniform: $C_1 = k_c = 1.0$, conservative. custom: your C_1 (1.0 to 2.8)

Frame mode uses the joint distribution factors of NCCI SN008 (the ENV 1993-1-1 Annex E method):

$$\eta = \frac{K_c + K_{col}}{K_c + K_{col} + \sum K_b}$$

with $K = I/L$. Non-sway frames use beam stiffness $0.5\,I/L$ (single curvature, conservative) and the non-sway L_{cr}/L expression; sway frames use $1.5\,I/L$ and the sway expression. You give lengths (and optionally picked sections) of the columns and beams above and below; a beam counts only once a section is picked, a missing length means no member, and a foundation base is taken as nominally rigid with the column's own stiffness (NCCI SN045). Non-sway K is limited to 0.5 to 1.0, sway K to 1 to 10; a sway mechanism is flagged.

4.3.3 Checks carried out

The calculation runs, for each load set:

1. **Classification** - EN 1993-1-1 Table 5.2 for web and flange, under the design N + M and under pure compression. Handles I, channel, RHS/SHS, tee, angle; CHS/EHS as tubes.
2. **Class 4** - effective area by a strip model (EN 1993-1-5 4.4) for I, channel, RHS/SHS, tee and single angle: A_{eff}, centroid shift e_N (additional moment $N \times e_N$, cl. 6.2.2.5(4)) and effective moduli. Class 4 CHS, EHS and back-to-back angles are **outside scope** and fail with that reason ("Class 4 section scope" row).
3. **Cross-section** - shear areas per cl. 6.2.6(3) (rolled/welded I, others), plastic shear resistance $V_{pl,Rd}$, shear buckling check when h_w/t_w is above the cl. 6.2.6(6) limit, moment resistance reduced for high shear (cl. 6.2.8, $\rho = (2V_{Ed}/V_{pl,Rd} - 1)^2$), axial compression $N_{c,Rd}$ (cl. 6.2.4, A_{eff} for class 4), axial tension $N_{t,Rd} = N_{pl,Rd}$ (cl. 6.2.3; net-section is not checked), bending with axial force (cl. 6.2.9: reduced plastic moments $M_{N,Rd}$ for class 1/2, linear interaction for class 3/4, with the cl. 6.2.9.1(4) exemption).
4. **Flexural buckling** about y-y and z-z - Table 6.1/6.2 curves, selected automatically from the section, thickness and grade (hollow: curve a, or a0 from S460; channels and tees: c; single angles: b; welded I: b/c or c/d).
5. **Torsional and torsional-flexural buckling** (cl. 6.3.1.4) where it applies, using I_w, I_t and the shear-centre offset for monosymmetric sections and angles ($N_{cr,TF}$ from the coupled equation).
6. **Lateral-torsional buckling** - for I sections the cl. 6.3.2.3 rolled/welded method with the UK NA values ($\lambda_{LT,0} = 0.4$, $\beta = 0.75$) and the f modification; the cl. 6.3.2.2 general case for other sections. M_{cr} follows NCCI SN003 over the z-z buckling length. The report notes when LTB can be ignored (cl. 6.3.2.2(4)).
7. **Member interaction** - Annex B (Method 2): Table B.2 for open sections, Table B.1 for hollow, class 1/2 or class 3/4 rows, equivalent uniform moment factors from Table B.3 ($C_m = 0.6 + 0.4\psi$, minimum 0.4, or 0.9 for sway). Expressions (6.61) and (6.62) give URB1 and URB2.

Governing utilisation is the maximum of: shear y-y and z-z, axial compression, axial tension, bending y-y and z-z, section interaction, buckling resistance in compression, LTB resistance, and combined buckling. Exactly 0 utilisation (an unloaded member) is a pass. The result is PASS when the governing utilisation is at most 1.00 and the section is not an unsupported class 4.

The summary card lists these rows (shear y-y, shear z-z, axial compression, axial tension, class 4 scope, bending y-y, bending z-z, biaxial bending, buckling in compression, buckling in bending, combined buckling), each with action, resistance, utilisation bar and PASS/FAIL. A **Working calculations** card shows every intermediate value (N_{cr}, λ, χ, α, k_{yy}, k_{zz}, M_{cr} and so on).

4.4 Reinforced concrete columns (EN 1992-1-1)

4.4.1 Inputs

Section size and the bar arrangement are edited on the section view; the rest are in the left panel.

Input	Default	Notes
Concrete class	C30/37 on the RC card; width $\times$ depth 300×300 mm	Classes C20/25 to C60/75
Reinforcement grade	B500B	B500A/B/C 500, B450C 450, B400 400 N/mm²
Bars	2 per face across b and across h (4 bars), 20 mm	Perimeter arrangement, corners shared: $n_{bars} = 2n_b + 2n_h - 4$, up to 12 per face

Input	Default	Notes
Links	8 mm at 175 mm	Legs for shear derived from the cage, or set
Exposure class	XC3/XC4	Sets $c_{min,dur}$
Design working life	50 years	50 or 100 (EC2 Table 4.3N structural class)
Special QC of concrete	off	-1 structural class (Table 4.3N)
Nominal cover	automatic	"Cover from exposure / fire" ON: $c_{nom} = c_{min} + 10$ mm rounded up to 5 mm. OFF: type it
$c_{min,dur}$ override	0 (automatic)	Otherwise the larger of BS 8500-1 Table A.4 and EC2 Table 4.4N at the Table 4.3N class
Maximum aggregate size	20 mm	Minimum bar gap = $\max(\text{bar dia}, d_g + 5, 20)$
Fire resistance	R60	Not checked, R30, R60, R90, R120, R180, R240
Fire load ratio μ_{fi}	0.7	Rounded up to the next Table 5.2a column (0.2, 0.5, 0.7)
Fire effective length $l_{0,fi}$	intermediate floor ($0.5\,l$)	Or top floor $0.7\,l$, or normal-temperature l_0
Braced	on	Off: unbraced (sway)
Effective length mode, each axis	factor	`factor` (F default 0.70), `stiffness` (relative flexibilities k_1, k_2 default 0.10, Exp. 5.15 braced / 5.16 unbraced), `direct` (type l_0)
End conditions	fixed-fixed	Y and Z
Slenderness and imperfection basis	conservative	See below
Creep coefficient $\varphi(\infty, t_0)$	2.0	
M_{0Eqp}/M_{0Ed}	0.70	Share of permanent moment for φ_{ef}
α_{cc}, γ_c, γ_s	0.85, 1.5, 1.15	UK NA values; $f_{yd} = 500/1.15 = 435$ N/mm^2
Mid-height first order moment	0	For members with transverse load
Take-off rebar from the design	on	Off: flat percentage allowance (default 2.0%) for carbon only

4.4.2 Calculation

For each axis and then for the combined section:

- **Imperfections and minimum eccentricity** - cl. 5.2 and 6.1(4): $e_i = \theta_i l_0/2$, with $\alpha_h = \max(2/3, \min(1, 2/\sqrt{l}))$. The conservative basis uses the cl. 5.2(9) simplification $e_i = l_0/400$ for braced columns and applies imperfections about both axes; the refined basis uses Exp. (5.2) and the single most unfavourable direction (cl. 5.8.9(2)). Minimum eccentricity $e_0 = \max(h/30, 20\text{ mm})$.
- **Slenderness** - $\lambda = l_0/i$ against the limit $\lambda_{lim} = 20\,A\,B\,C/\sqrt{n}$ (cl. 5.8.3.1). Conservative basis: $A = 0.7$; refined: $A = 1/(1 + 0.2\varphi_{ef})$. $B = \sqrt{1 + 2\omega}$. $C = 1.7 - r_m$ with a floor of 0.7; $C = 0.7$ for transverse load or unbraced columns. r_m uses the end moments with the imperfection moment added (conservative basis).
- **Second order** - if slender, nominal curvature (cl. 5.8.8): $1/r = K_r K_\varphi / r_0$, with $c = 10$, K_r from n and $n_{bal} = 0.4$, K_φ from $\beta = 0.35 + f_{ck}/200 - \lambda/150$, and d taken as $h/2 + i_s$ where bars are not all on two faces (Exp. 5.35).
- **Design moment** - M_{02} plus imperfection (floored at the minimum eccentricity moment), $M_{0Ed} = \max(0.6\,M_{02} + 0.4\,M_{01},\ 0.4\,M_{02})$, then $M_{Ed} = \max(M_{02},\ M_{0Ed} + M_2,\ M_{01} + M_i + 0.5\,M_2)$ for braced columns; $M_{02} + M_2$ for unbraced; the mid-height section for transverse load.

- **Section resistance** - strain-compatibility with the rectangular stress block (strains per EC2 Figure 6.1) for the actual bar positions, giving M_{Rd} for the axial load, an N-M interaction diagram, and the reinforcement area required.
- **Biaxial bending** - cl. 5.8.9: separate checks are accepted when the slenderness and eccentricity ratios satisfy Exp. (5.38a/b); otherwise the Exp. (5.39) interaction with exponent a (from N_{Ed}/N_{Rd}) governs. $N_{Rd} = A_c f_{cd} + A_s f_{yd}$.
- **Axial tension** - carried by the bars alone: $N_{Rd,t} = A_s f_{yd}$.
- **Shear** - cl. 6.2.2 and 6.2.3 with axial force: $V_{Rd,c}$ with the UK NA treatment (shear strength of classes above C50/60 taken as C50/60; $\alpha_{cc} = 1.0$ for shear), links by the variable strut method with $1 \le \cot\theta \le 2.5$ (1.25 in tension), $V_{Rd,max}$ cap. Links resist shear on the perimeter legs parallel to the shear.
- **Detailing** - cl. 9.5.2/9.5.3: $A_{s,min} = \max(0.10\,N_{Ed}/f_{yd},\ 0.002\,A_c)$; $A_{s,max} = 0.04\,A_c$; minimum bar diameter 12 mm (UK NA; EC2 recommends 8); at least 4 bars; link diameter at least $\max(6,\ 0.25 \times \text{bar dia})$ (the report also notes the 8 mm good practice); link spacing at most $\min(20 \times \text{bar dia},\ \text{least dimension},\ 400\ \text{mm})$, reduced by 0.6 within the zones of cl. 9.5.3(4) (near beams/slabs and laps), which the tool applies to the whole lift, so the check uses $0.6\times$ that limit; compression-zone bar-to-tied-bar distance 150 mm (cl. 9.5.3(6)); a section deeper than four times its width is flagged as a wall (cl. 9.5.1(1)).
- **Cover** - cl. 4.4.1 with BS 8500-1: c_{nom} is checked against $c_{min} + 10\ \text{mm}$. A user cover below the bond or fire minimum fails; below only the BS 8500 durability value gives "Review required" (a flag, not a fail). The concrete class is checked against the BS 8500 minimum for the exposure.
- **Fire** - EN 1992-1-2 Table 5.2a (Method A, braced columns exposed on more than one side): required minimum dimension b_{min} and axis distance a for the period and μ_{fi}. Validity conditions ($l_{0,fi} \le 3\ \text{m}$, $e_{Fi} \le 0.15\,h$, $A_s \le 4\%\,A_c$, bar count) are checked; outside them the row shows "CHECK B" (use Table 5.2b Method B or Annex C) and the check fails until that is done. The card also explains the fix (larger column, more axis distance, justify a lower μ_{fi}).

Summary rows (concrete): axial compression (or tension), bending + axial y-y and z-z, biaxial bending, shear y-y and z-z, reinforcement area (required vs provided), detailing cl. 9.5, nominal cover, and fire resistance. The column passes when governing utilisation is at most 1.00 **and** detailing, cover and fire all pass. A required steel area beyond about 64 times the provided area is shown as "more than the section can carry".

Further cards: **Slenderness and second order effects**, **N-M interaction diagram** (the section's envelope with the design points), **Concrete column properties**, **Column section + 3D views** (live reinforcement), **Optimised column options** (an on-demand search for alternative size and bars) and **Working calculations**.

4.5 Quantities and carbon

Steel: mass = Blue Book kg/m × length × quantity. RC: concrete volume and the rebar mass measured off the designed cage (bars and links), or the flat allowance. Embodied carbon (A1-A3) uses factors defaulted from IStructE Table 2.5 (steel industry mean; RC defaults concrete 217 and rebar 0.76 $kgCO_2e$ per unit, with named factor choices, custom factors and EPD file fields). A4 (default 12%) and A5 (default 4%) are percentages of A1-A3. A bar-bending-schedule override is available. The same quantities feed the carbon totals elsewhere in the workspace. Quantity (number of identical columns) defaults to 1.

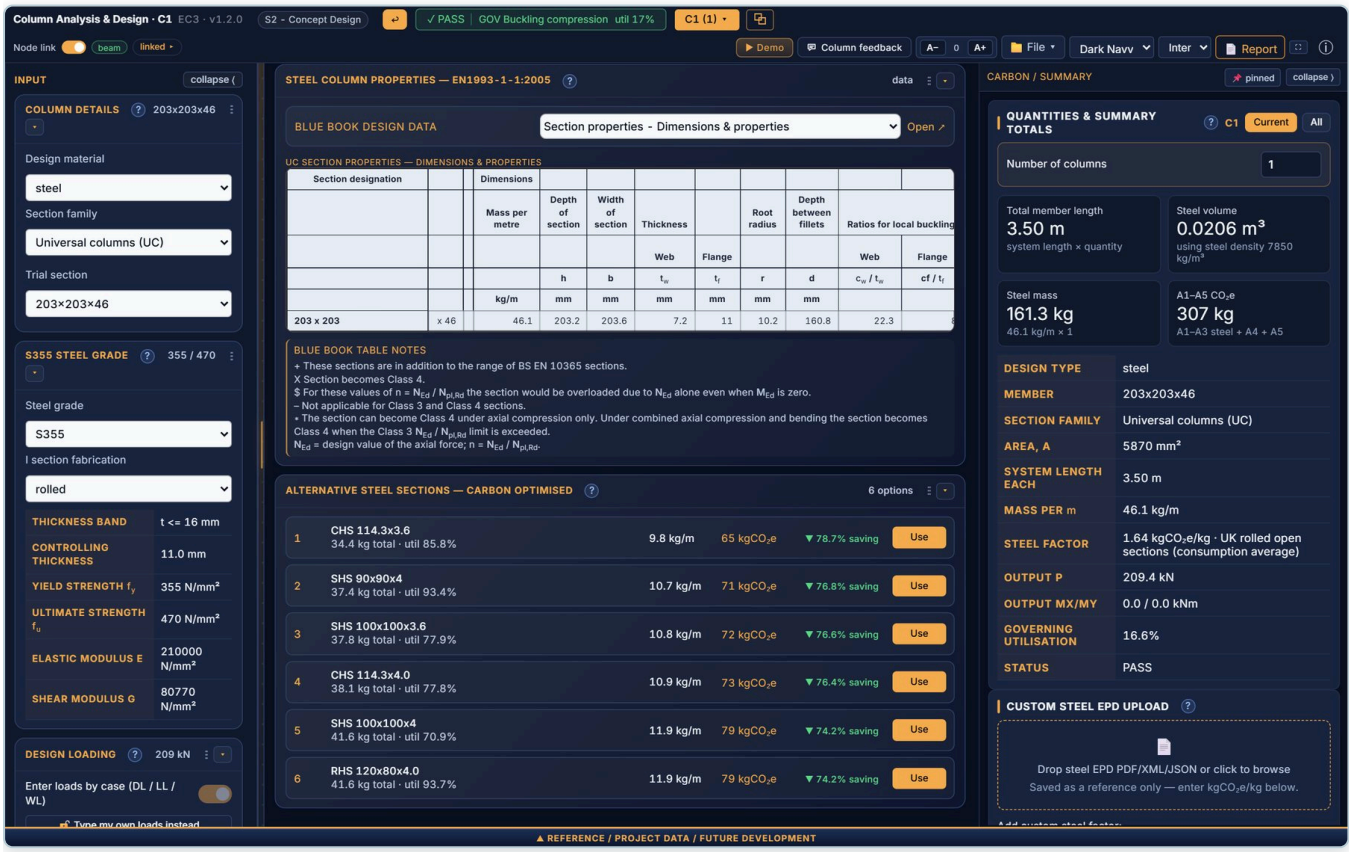
4.6 Results and reports

- A header status chip and the node card show governing utilisation and PASS/FAIL. Utilisation colours: pass up to 85%, amber 85 to 100%, fail above 100%.
- **Report**: Word (`..._column_design_report.doc`), PDF (opens the print dialogue, choose Save as PDF), copy, or text (`..._column_design_calc.txt`). The window offers four colour schemes (Amber engineering, Black / white, Corporate blue, Reading comfort) and re-reads the live design every time it is drawn (Regenerate). It follows the shared report format used by all modules: summary, load basis, working sections, views, carbon, references.
- The load section states plainly the basis: for "design" rows it says the rows are summed with no γ, ψ_0 factors applied; for "characteristic" rows it shows the governing set and how each of N_{Ed}, V_{Ed}, M_{Ed} was formed.
- Standards referenced in the report: steel - EN 1993-1-1:2005+A1:2014 (Table 3.1, 5.5/Table 5.2, 6.2.4 to 6.2.9, 6.3.1 to 6.3.3), UK NA, SCI P363 / Blue Book. Concrete - EN 1992-1-1:2004+A1:2014 (3.1.6, 3.1.7, 5.2, 5.8.3 to 5.8.9, 6.1 to

6.2.3, 9.5.2, 9.5.3), EN 1992-1-2 Table 5.2a, BS 8500-1, UK NA, and The Concrete Centre guidance on columns.

4.7 Alternative-section searches

The steel and RC "alternatives" cards never run on their own. Press the card's run button; results remain while inputs are unchanged, and a note says "inputs changed" when they are not current.



Screenshot: A steel column with the Blue Book section data and the carbon-optimised alternative sections list; Use replaces the trial section.

4.8 Connections to other nodes

Outputs

To	What is sent
Foundation (pad or pile cap only)	Per case DL, LL and WL for P, V_x, V_y, M_x, M_y (or DLz etc. from direct input); the column self-weight added to DLz; column width and depth (for steel, taken from the section dimensions; RC: concrete size); section/grade text; utilisation; mass and carbon. Each item can be switched off on the wire (fields P, V_x, V_y, M_x, M_y, size, section, utilisation, carbon) and the target can be one foundation or all eligible ones
Base plate	Design actions after the column's own factoring, the section and grade. Per-wire options choose cases (DL, LL, WL, self-weight SW), shear source (Vx, Vy or resultant) and moment source (Mx, My or resultant), plus manual add-ons. See Chapter 5
Math	Values for formulas

The foundation receives characteristic rows case by case ("still sends the unfactored rows"), so the foundation applies its own factors. An identical send is skipped: a wire remembers what it delivered last and skips an identical send; a manual refresh on the wire always sends.

Inputs: see 4.2.3, plus Blue Book (section only).

4.9 Tips, limitations and caveats

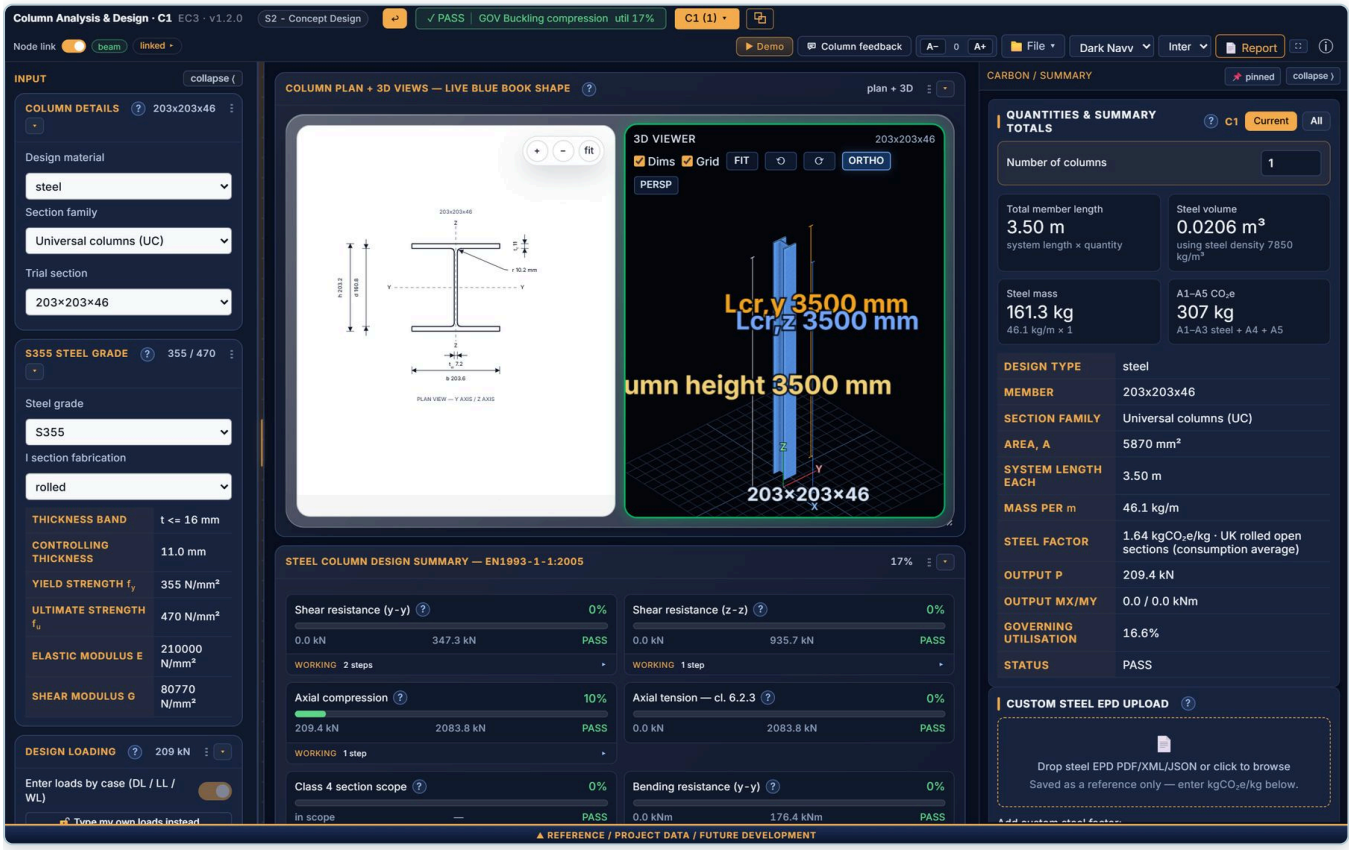
- Check the "The row values are" selector. Mixing factored values into a characteristic grid (or the reverse) is the most common error.
- With a wind row, the RC check can be sensitive to the lower-axial pairing. The $1.0G$ sets are included, but if the wind case relieves the axial load, also confirm the result in your own review.
- Steel moments about each axis use the end moments for the ratio ψ, so give both bottom and top moments; leaving the top at 0 treats the column as having a single-ended moment.
- Class 4 CHS/EHS and built-up members are not designed.
- Net-section tension ($N_{u,Rd}$) is not checked for steel.
- RC columns are rectangular only, with a perimeter bar arrangement. RC fire is Method A only; outside its limits you must use Method B or Annex C outside the tool.
- RC α_{cc} defaults to 0.85 (UK NA). The EN recommended 1.0 is available by editing the field.
- Column self-weight is not included in the column's own check, only in what it sends on.
- The default section sizes and loads are starting values, not recommendations.

> **References.** The standards, guidance and data sources used in this chapter are listed, with web links, in Chapter 17.

Screen reference - panel by panel

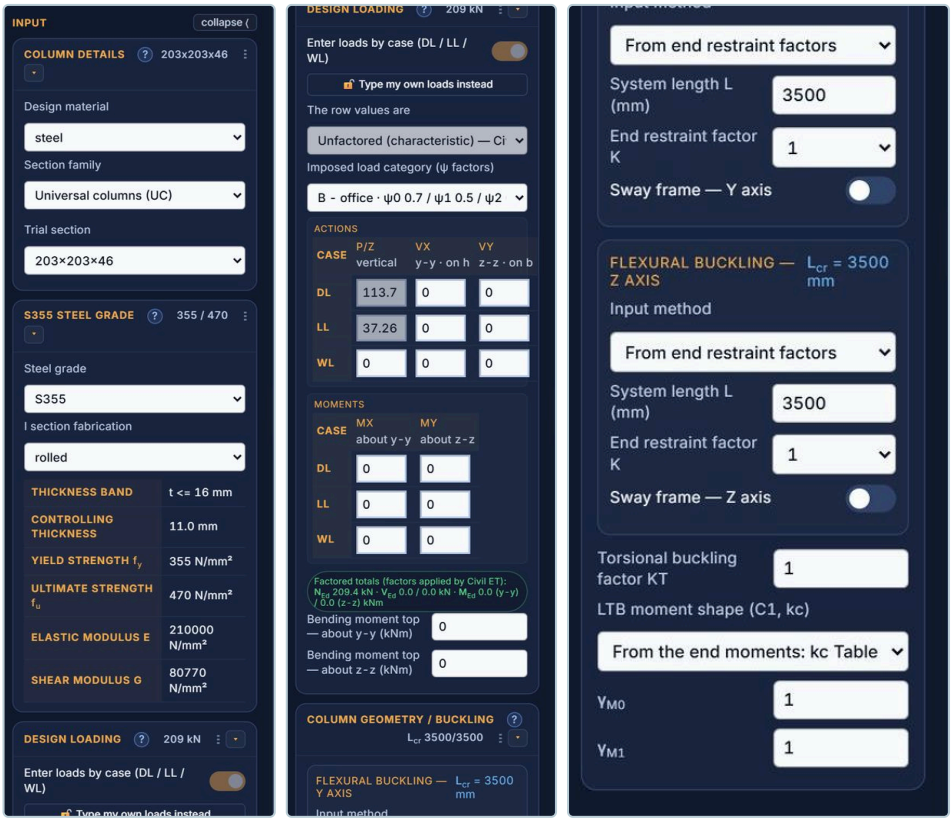
These screenshots show every panel of the module exactly as it appears on screen, scrolled from top to bottom, so you can match any field, button or result to the explanations in this chapter. Cards that are collapsed by default are shown expanded.

Steel column: the whole screen



Steel column: the whole screen.

Steel column: input panel (left of the screen)

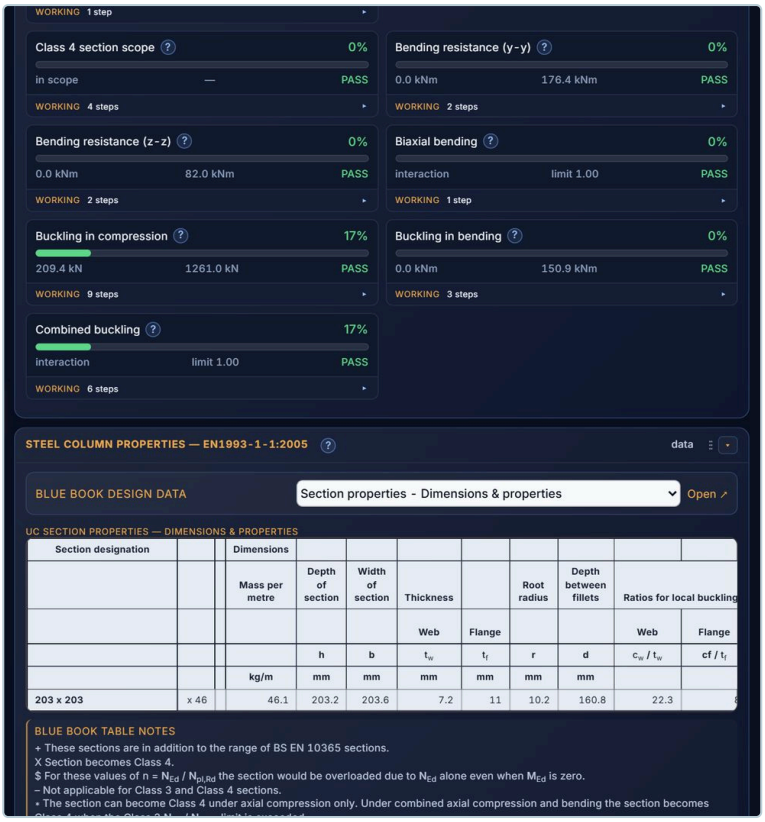


Steel column, input panel (left of the screen), parts 1-3 of 3. Cards shown: Column details, S355 steel grade, Design loading, Actions, Moments, Column geometry / buckling, Flexural buckling — y axis, Flexural buckling — z axis.

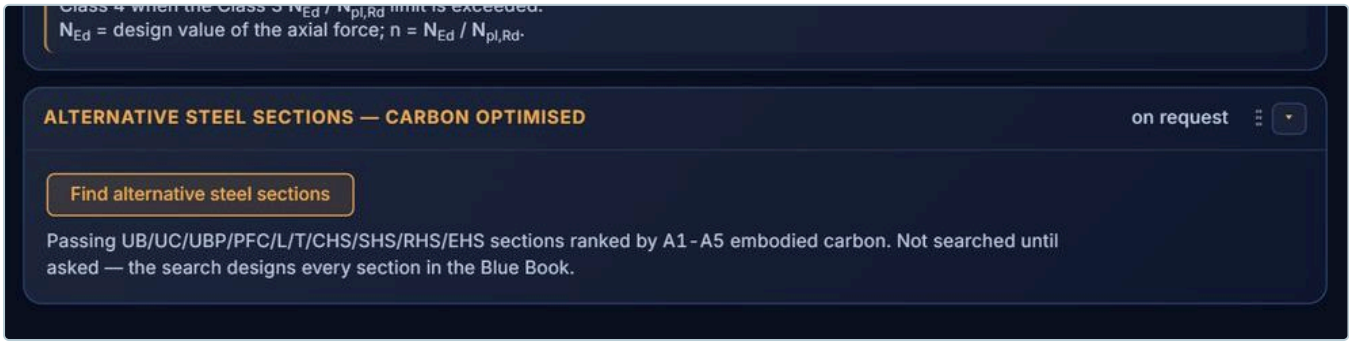
Steel column: working panel (centre of the screen)



Steel column, working panel (centre of the screen), part 1 of 3. Cards shown: Column plan + 3D views — live Blue Book shape, 3D viewer, Working.

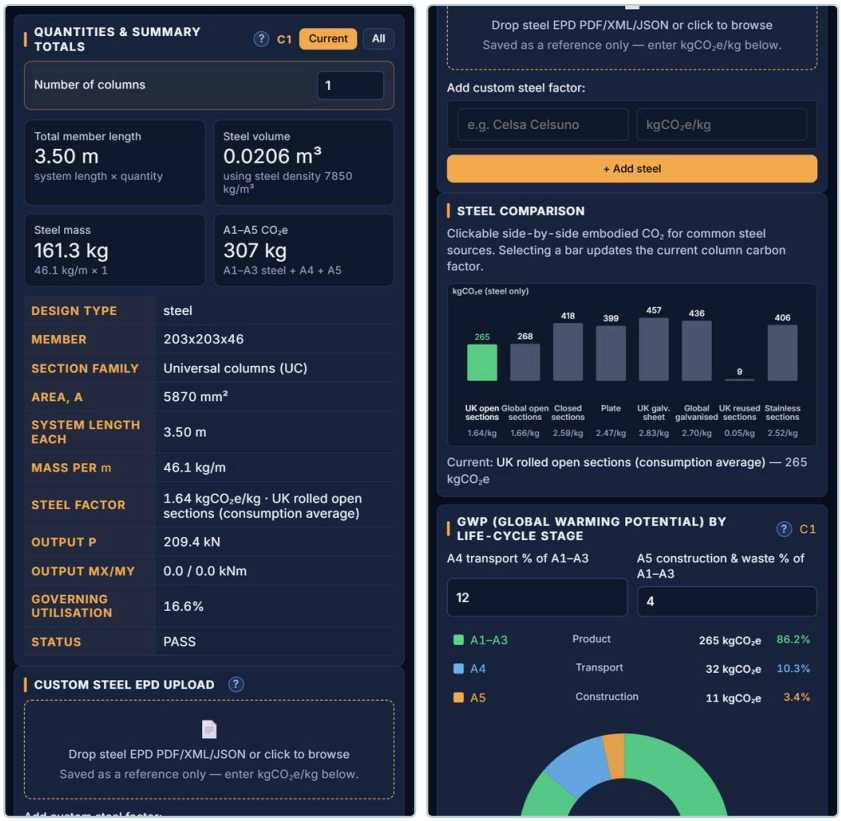


Steel column, working panel (centre of the screen), part 2 of 3. Cards shown: Working.



Steel column, working panel (centre of the screen), part 3 of 3. Cards shown: Working.

Steel column: results panel (right of the screen)

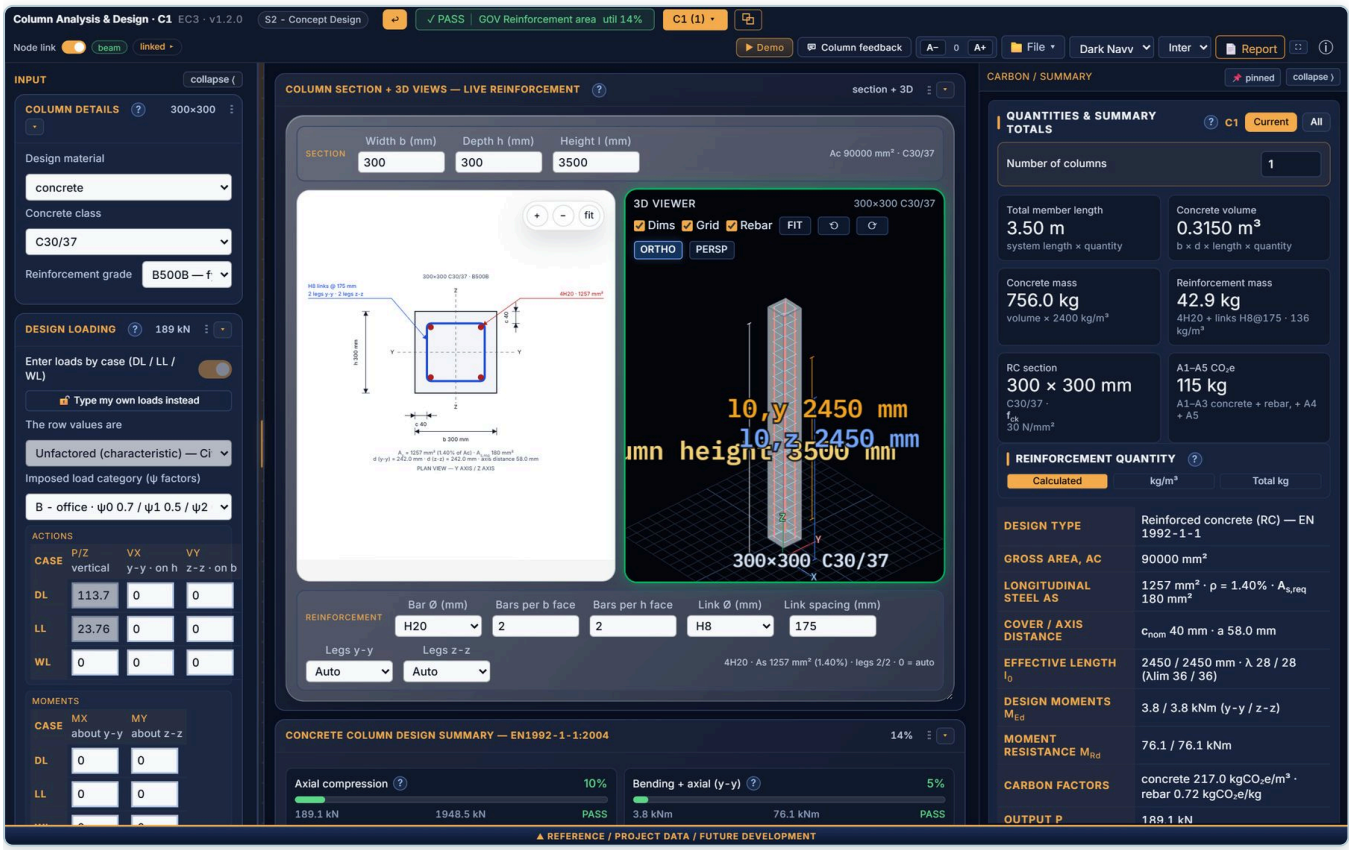


Steel column, results panel (right of the screen), parts 1-2 of 3. Cards shown: Custom steel EPD upload, Steel comparison, GWP (global warming potential) by life-cycle stage C1.



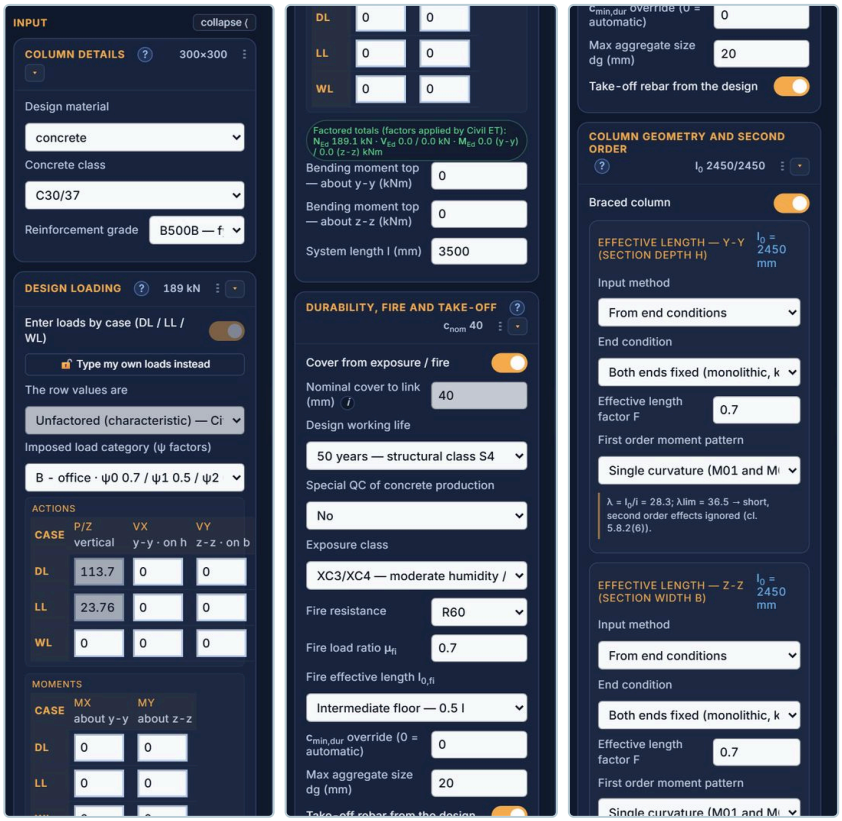
Steel column, results panel (right of the screen), part 3 of 3. Cards shown: GWP (global warming potential) by life-cycle stage C1.

Reinforced concrete column: the whole screen

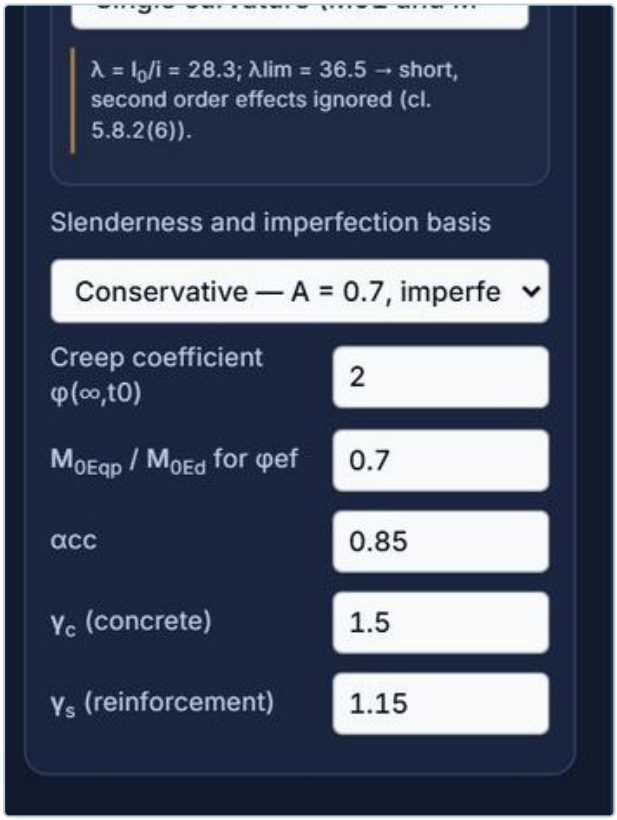


Reinforced concrete column: the whole screen.

Reinforced concrete column: input panel (left of the screen)



Reinforced concrete column, input panel (left of the screen), parts 1-3 of 4. Cards shown: Column details, Design loading, Actions, Moments, Durability, fire and take-off, Column geometry and second order, Effective length — y-y (section depth h), Effective length — z-z (section width b).

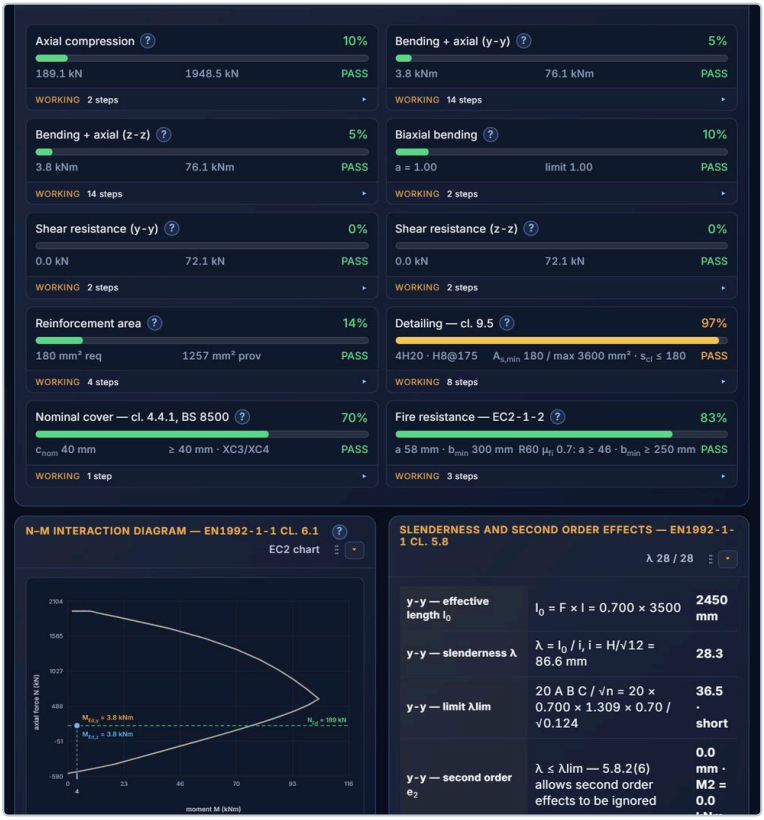


Reinforced concrete column, input panel (left of the screen), part 4 of 4. Cards shown: Effective length — z-z (section width b).

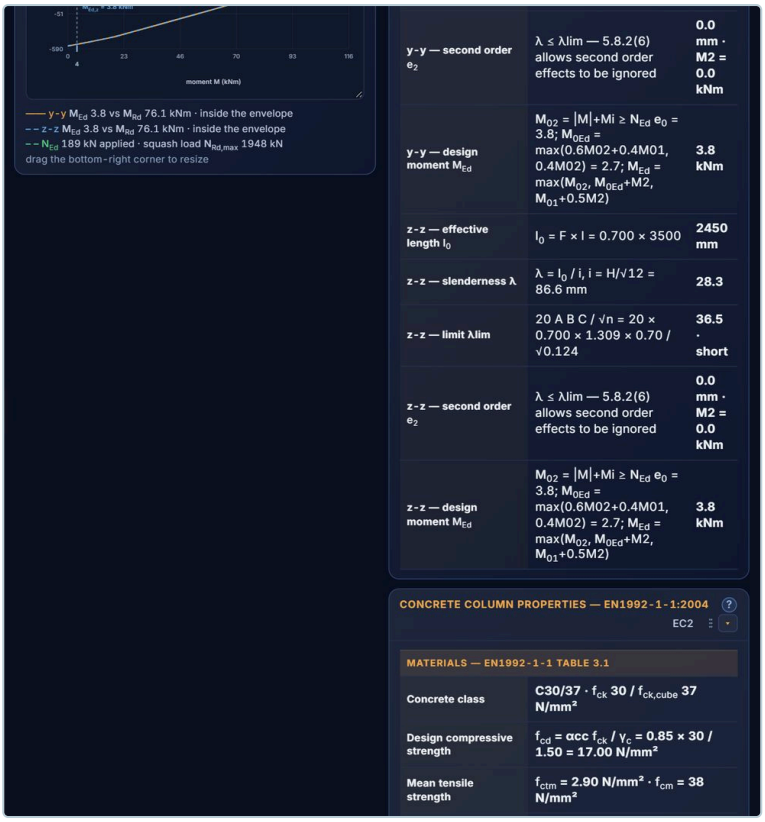
Reinforced concrete column: working panel (centre of the screen)



Reinforced concrete column, working panel (centre of the screen), part 1 of 5. Cards shown: Column section + 3D views — live reinforcement, 3D viewer.



Reinforced concrete column, working panel (centre of the screen), part 2 of 5. Cards shown: Working.



Reinforced concrete column, working panel (centre of the screen), part 3 of 5.

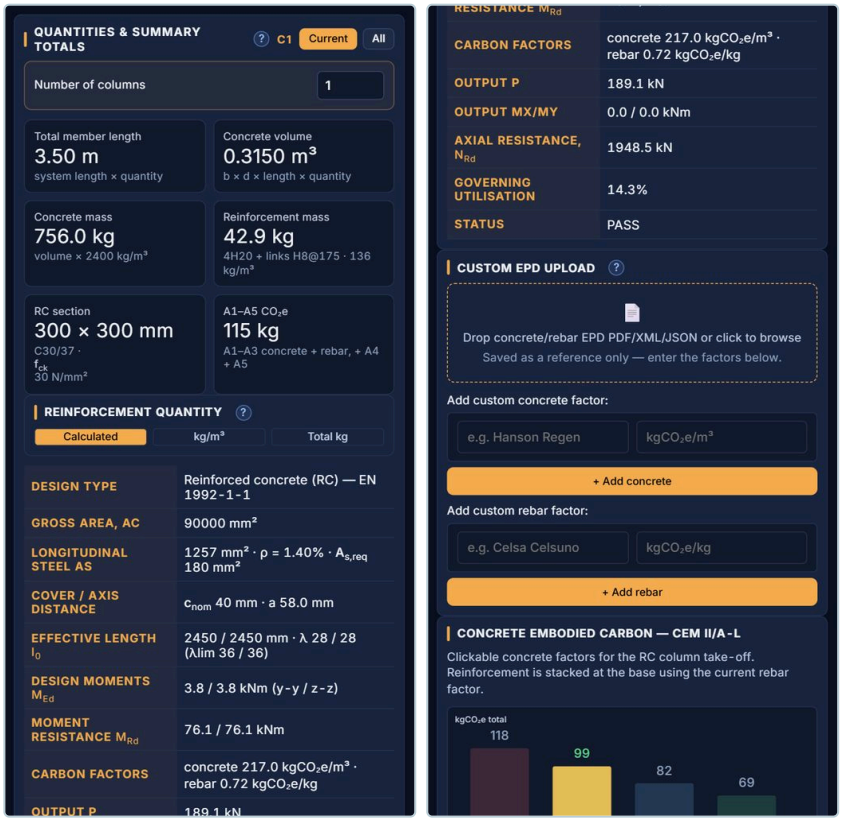


Reinforced concrete column, working panel (centre of the screen), part 4 of 5.



Reinforced concrete column, working panel (centre of the screen), part 5 of 5. Cards shown: Optimised column options.

Reinforced concrete column: results panel (right of the screen)



Reinforced concrete column, results panel (right of the screen), parts 1-2 of 4. Cards shown: Reinforcement quantity, Custom EPD upload, Concrete embodied carbon — CEM II/A-L.



Reinforced concrete column, results panel (right of the screen), parts 3-4 of 4. Cards shown: Reinforcement comparison, GWP (global warming potential) by life-cycle stage C1.

CHAPTER 5

Base plate and anchor design

The Base plate node checks a steel column base: a plate welded to a UC (or other section from the Blue Book list) and held down by anchor bolts into a concrete foundation. It uses the component method of EN 1993-1-8 (T-stubs for the plate in tension, a bearing T-stub on the concrete, Table 6.7 for the moment resistance) and, if you tick the option, a full anchor check to ETAG 001 Annex C / EOTA TR029 (steel, pull-out, concrete cone, pry-out, combined tension and shear). It also checks the column-to-plate weld and the transfer of shear by friction plus the anchors.

The node is marked Beta in the node library. It uses the loads it is given as final design actions: it applies no partial factors to N_{Ed}, V_{Ed} or M_{Ed}.

5.1 Opening and adding a base plate

- Base plate is a core node (alongside Foundation and Column). Others can be added from the library (tag "EC3 / 1-8").
- Like the Column node, one Base plate node holds several designs (BP1, BP2, ...), each with its own dots on the node card showing the utilisation. Open the node from the map to get the workspace: an input panel on the left (collapsible and resizable), results and workings on the right, and a header picker.
- The input panel has these cards: Design forces, Column details, Base plate details, Anchor layout, Foundation / concrete, and (when ETAG design is on) ETAG 001 / TR029 anchor data.

5.2 Choosing a calculation basis

A "Calculation basis" switch in the Design forces card selects how the concrete bearing strength is found:

Basis	Behaviour
EN 1993-1-8 detailed (default)	Iterates the joint bearing strength f_{jd} from the loaded area A_{c0} and the maximum distribution area A_{c1} (EN 1993-1-8 6.2.5 with EN 1992-1-1 6.7). Up to 20 iterations; the additional bearing width c is recomputed each pass
SCI P398 simplified	Section 5 / Appendix E hand-check route: $\alpha = 1.5$ and $\beta_j = 2/3$, so $f_{jd} = \beta_j \times \alpha \times f_{cd} = f_{cd}$. No A_{c0}/A_{c1} loop. Best for transparent manual verification

The rest of the calculation is common to both bases. (In older builds this switch was labelled with a third-party software name; v1.2.0 renamed the styles, see 5.9.)

5.3 Inputs

5.3.1 Design forces

Input	Default	Notes
Use column node data	off	ON: forces and column section come from a connected Column node (see 5.7)
Include N_{Ed} / V_{Ed} / M_{Ed}	all on	Switch off to isolate checks (for example axial/shear only)
N_{Ed} (kN)	350	Compression design axial load. Negative values are treated as zero: the node designs for compression plus the moment's tension side
V_{Ed} (kN)	75	Base shear
M_{Ed} (kNm)	100	Base moment (magnitude is used)

When the column wire is active the three fields are disabled and show the transferred values, with a summary line ("DL+LL+WL -> N, V, M"). A "Restore manual" button returns to the numbers you typed before linking.

5.3.2 Column details

Input	Default	Notes
Column section	305x305x118	Chosen from the UC list (disabled when linked)
Column steel grade	S355	From the standard grade list (not "User defined")
Eccentricity $e_{bp,x}$ (mm)	0	Offset of the column on the plate
Weld leg size s_w (mm)	12	6, 8, 10, 12 or 14 mm. Throat $a = s/\sqrt{2}$

5.3.3 Base plate details

Input	Default	Notes
Plate length h_p (mm)	600	Minimum 100 used in calculation
Plate width b_p (mm)	600	
Plate thickness t_p (mm)	40	Limited to 10 to 150 mm
Plate grade	S275	Plate grade list (S235 to S690Q); "User defined" adds f_y and f_u (275 / 410). f_y, f_u follow the thickness band of the plate
Friction coefficient $C_{f,d}$	0.20	EN 1993-1-8 6.2.2(6)
Include friction check	on	Turn off, or set the coefficient to 0, when the anchors are to carry all the shear

5.3.4 Anchor layout

Input	Default	Notes
Design anchors using ETAG 001 / TR029	on	See 5.5
Anchor diameter d_{nom} (mm)	24	12, 16, 20, 22, 24, 27, 30, 33, 36
Bolt hole clearance (mm)	2	1, 1.5, 2, 3, 4, 6
LHS bolts each row, RHS bolts each row	2 and 2	ETAG design is limited to 3 per side; a layout beyond that is flagged and forces the result to fail (utilisation not below 1.01)
LHS / RHS edge x, edge y (mm)	50 each	Edge distances for the left and right bolt rows (x and y); x also sets the tension-side lever arm from the plate edge

The two sides (left and right of the column about the bending axis) each carry n bolts in a row across the plate width; the tension side is the left-hand row n_1 in the formulas.

5.3.5 Foundation / concrete

Input	Default	Notes
Use foundation node data	off	ON: thickness, plan size and concrete class come from a connected Foundation
Foundation length L, width B (mm)	3000 × 3000	
Foundation thickness t_{fnd} (mm)	700	Also feeds the bearing distribution width on the tension side
Concrete class	C25/30	C20/25, C25/30, C30/37, C32/40, C35/45, C40/50. $f_{cd} = 0.85 f_{ck}/1.5$ ($\alpha_{cc} = 0.85$, $\gamma_c = 1.5$, fixed in this node)

5.3.6 ETAG 001 / TR029 anchor data (shown when ETAG design is on)

Group	Input	Default
Anchor details	Anchor name, type (Bonded or Mechanical)	blank, Bonded
	f_{yk}, f_{uk} (N/mm²)	640, 800
	Effective anchorage depth h_{ef} (mm)	250
	Effective length under shear l_f (mechanical only)	120
Splitting due to installation	s_{min}, c_{min}, h_{min} (mm)	120, 120, 176
Splitting due to loading	$s_{cr,sp}$, $c_{cr,sp}$ (mm)	300, 150
Pry-out	Calculate pry-out factor	on ($k_{pry} = 2$ when $h_{ef} \geq 60$ mm, else 1; or choose 1 or 2)
Steel failure	Calculate tension / shear resistance	both on (otherwise enter $N_{Rk,s}$ 282 kN, $V_{Rk,s}$ 141 kN)
Pull-out	$\tau_{Rk,cr}$ / $\tau_{Rk,ucr}$ (bonded)	8 / 14 N/mm²
	$N_{Rk,p}$ (mechanical) and increasing factor ψ_c	150 kN, 1.0
Concrete	Cracked or Non-cracked	Cracked
	Shell spalling reinforcement	off
	Reinforcement to control splitting	on
	Edge reinforcement type	No edge reinforcement
Factors	Installation $\gamma_{2,t}$	1.2 (1.0, 1.2 or 1.4)
	Shear installation $\gamma_{2,s}$	1.0
	Combined interaction α	1.5 (1.0, 1.5 or 2.0)

Shell spalling: tick when the anchorage-zone reinforcement meets the TR029 spacing rule (given in the field note); otherwise $\psi_{re,N} = 0.5 + h_{ef}/200$ (maximum 1).

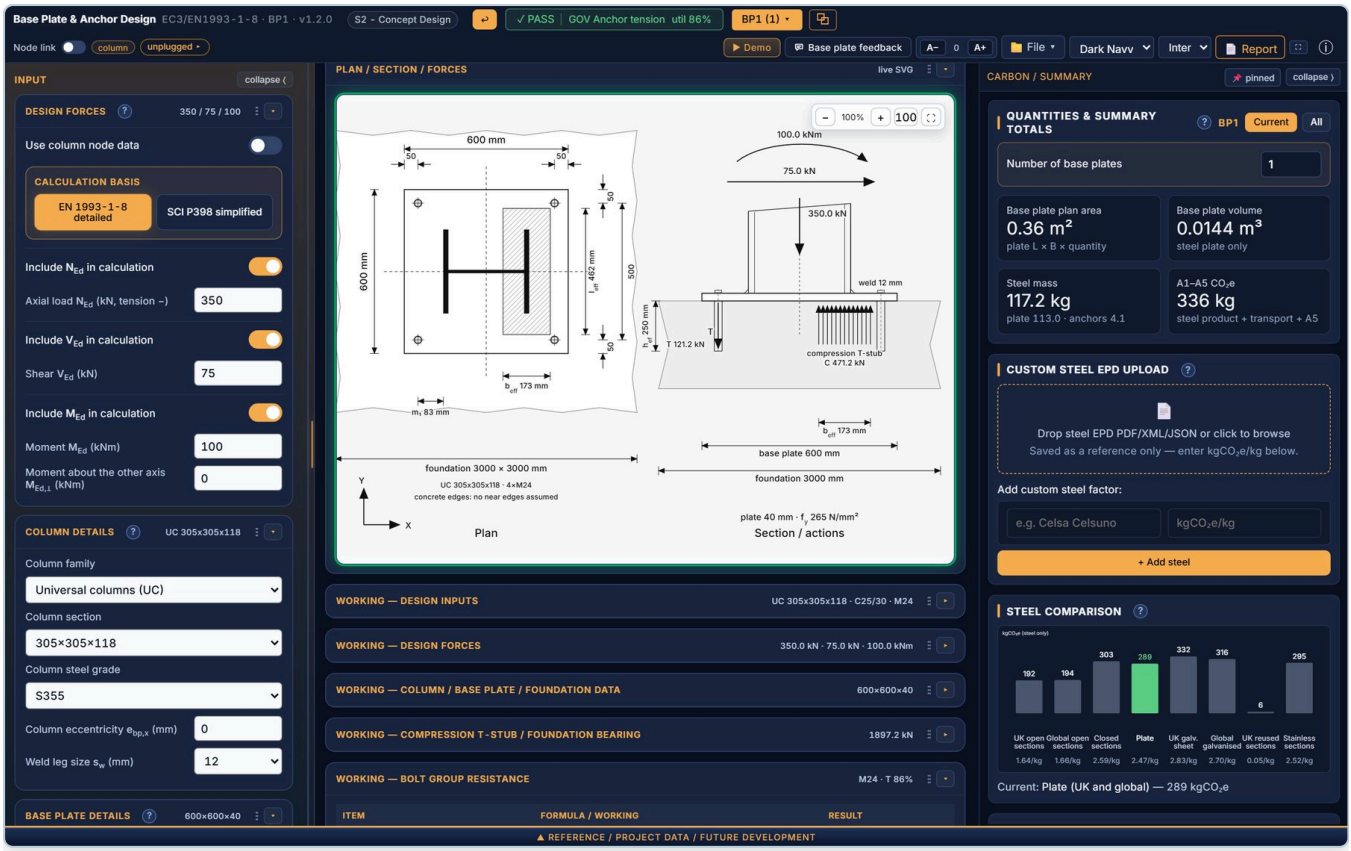
If ETAG design is off, the anchors are treated as ordinary bolts: tension $0.9 f_{uk} A_s/1.25$ and shear $k_6 f_{uk} A_s/1.25$, with the stress area from the diameter.

5.4 What the calculation does

Principal steps:

1. **Geometry.** Lever arms from the column section: $z_{C,l} = z_{C,r} = (D - T)/2$ (flange centres). Tension-side lever arms $z_{T,l}$, $z_{T,r}$ from the plate length, eccentricity and bolt edge distance. $z_{C,C} = z_{C,l} + z_{C,r}$; $z_{T,C} = z_{T,l} + z_{C,r}$.

2. **Branch selection (EN 1993-1-8 Table 6.7).** The compression-compression branch is used while both flange T-stub forces stay compressive, i.e. $N_{L,cc}$ and $N_{R,cc}$ computed from N_{Ed}, M_{Ed} and $z_{C,C}$ are non-negative. Otherwise the tension-compression branch applies. The tension-side test uses $N_{Ed,T}$ (see 5.4.1), the compression side the full N_{Ed}.
3. **Bearing on the concrete (EN 1993-1-8 6.2.5).** Additional bearing width $c = t_p\sqrt{f_y/(3f_{jd}\gamma_{M0})}$, the effective T-stub dimensions b_{eff} and l_{eff}, and the compression resistance $F_{C,Rd} = f_{jd}\,b_{eff}\,l_{eff}$. This is capped by the column flange's resistance, $M_{c,Rd}$ divided by the flange lever arm $(D - T)$. Axial resistance for pure compression is $f_{jd} \times 0.945 \times h_p \times b_p$.
4. **Tension T-stub (EN 1993-1-8 6.2.6).** Effective length l_{eff} is the smaller of the circular and non-circular patterns for the bolt row (formulas with m, e, p), $M_{pl} = 0.25\,l_{eff}\,t^2 f_y/\gamma_{M0}$, and the mode 1-2 resistance $F_{T,1-2,Rd} = 2M_{pl}/m$. Mode 3 is the bolt group $n \times F_{t,Rd}$. $F_{tPl,Rd}$ is the lower of the two. Mode 3 is active in every route, so the anchor diameter always affects the result.
5. **Moment resistance (Table 6.7).** In the compression-compression branch $M_{j,Rd}$ is the lesser of the left and right limits; in the tension-compression branch it is the lesser of the tension limit ($F_{tPl,Rd}$ with the axial benefit) and the compression limit. A zero moment gives infinite resistance; the check on moment is replaced by the axial check.
6. **Weld.** Fillet weld to the plate: $f_{w,Rd}$ uses $f_{vw} = \min(f_{u,\text{column}}, f_{u,\text{plate}})/(\sqrt{3} \times \beta_w \times \gamma_{M2})$ with $\beta_w = 0.85$ and $\gamma_{M2} = 1.25$, times the throat. Tension weld length $L_{wt} = 2B - (t_w + 2r)$ (flange welds; EN 1993-1-8 4.5.3). Shear weld length $L_{wv} = 2(D - 2(T + r))$ (the web). Weld tension demand is $|M|/(D - T) - N_{Ed}/2$, and applies only in the tension-compression branch.
7. **Shear transfer.** Friction $F_{f,Rd} = C_{f,d} \times N_{c,Ed}$, where $N_{c,Ed}$ is the column's design compression (the minimum coexisting value, $N_{Ed,T}$). Anchors carry $V_{Ed} - F_{f,Rd}$, floored at zero. The check is $V_{Ed}/(F_{f,Rd} + \text{anchor shear resistance})$, per EN 1993-1-8 6.2.2(7). Friction alone below V_{Ed} is therefore not a failure while the anchors carry the rest. Previously the compression-block force (N plus anchor pull) was used for friction, which counted anchor tension as clamping force; that was corrected in version 1.2.0.
8. **Anchors (ETAG on).** For the tension row (n_1 anchors):
   - steel: $N_{Rk,s} = A_s f_{uk}$, $\gamma_{Ms} = \max(1.2/(f_{yk}/f_{uk}), 1.4)$;
   - pull-out: bonded $N_{Rk,p} = \pi\,d\,h_{ef}\,\tau \times A_{pN}/A^0_{pN} \times \psi$ factors, $s_{cr,Np} = \min(20\,d\sqrt{\tau_{ucr}/7.5},\ 3h_{ef})$; mechanical uses the entered single-anchor value;
   - concrete cone: $N^0_{Rk,c} = k_1\sqrt{f_{ck,cube}}\,h_{ef}^{1.5}$, $k_1 = 7.2$ cracked or 10.1 non-cracked, with A_{cN}/A^0_{cN} ($s_{cr,N} = 3h_{ef}$);
   - splitting is entered as always satisfied (no resistance applied in the calculation);
   - shear: steel failure without lever arm $V_{Rk,s} = k_6 A_s f_{uk}$ (EN 1992-4 (7.35)), $k_6 = 0.6$ for $f_{uk} \le 500$ N/mm² and 0.5 above (an 8.8 anchor is 0.5; the EN 1993-1-8 bolt value of 0.6 is not used for ETAG anchors); $\gamma_{Ms,V} = \max(1.0/(f_{yk}/f_{uk}), 1.25)$; pry-out $V_{Rk,cp} = k_{pry} \times N_{Rk,c}$ over the pry-out area;
   - partial factors $\gamma_{Mc} = \gamma_c \times \gamma_2$ (tension) and $\gamma_c \times \gamma_{2,s}$ (shear);
   - combined: $\beta_N^\alpha + \beta_V^\alpha$, where β_N is the largest of steel, pull-out and cone ratios, and β_V the larger of steel and pry-out.
9. **Governing utilisation.** With ETAG on: maximum of moment, compression, weld tension (when the tension branch is active), weld shear, anchor tension (when active), anchor shear, combined anchor. With ETAG off: moment, compression, weld tension and shear, bolt tension (when active), bolt shear (when needed) and the shear-transfer check. The ETAG layout limit (more than 3 per side) forces a fail.
10. **Quantities.** Plate mass = $h \times b \times t \times 7850$ kg/m³; anchor mass from the number of anchors, diameter and length (h_{ef} + plate thickness). Carbon A1-A3 uses factors defaulted to 1.74 kgCO_2e per kg for plate and anchors (editable); A4 12% and A5 4% by default.



Screenshot: The base plate module: design forces and calculation basis on the left, the plan, section and forces drawing in the middle.

5.4.1 Tension-side axial: NEd,T

Anchor tension is governed by low axial load with high moment. When the column wire carries characteristic rows (so the (6.10) envelope applies), the wire also carries $N_{Ed,min}$, the least axial of the two complete (6.10) combinations. The base plate pairs the envelope moment with this least axial on the tension side (anchor tension, mode tests, friction) and keeps the full N_{Ed} on the compression side. In design mode and for manual inputs $N_{Ed,T}$ equals N_{Ed}, so nothing changes.

5.5 Results

The summary card lists the checks with action, resistance, utilisation bar and PASS/FAIL. With ETAG on the rows are: Moment (or Axial when M_{Ed} is zero), Shear transfer (friction + anchors), then, if the tension side is active, Weld tension, Anchor tension, Bolt group tension; then if anchor shear exists, Anchor shear and Combined anchor loads. With ETAG off: Moment/Axial, Shear transfer, Weld tension and Bolt group tension (tension side), Bolt shear, Weld shear. Tension rows appear only when the anchors are actually in tension.

The header and node card show the governing check by name and the utilisation. Utilisation colours: pass up to 85%, amber from 85% to 100%, fail above 100%.

Working cards (each can be collapsed and re-ordered): Plan / section / forces (live SVG), Design forces, Column / base plate / foundation data, Design forces in T-stubs, Compression T-stub / foundation bearing, Tension T-stub / base plate bending, Bolt group resistance, Moment resistance, Weld resistance, ETAG 001 / TR029 anchor resistance (if on), and Steel quantity and carbon. Every intermediate figure is printed in the working tables.

5.6 Report

Open the Report from the node toolbar. Choices are Word (`..._base_plate_design_report.doc`), PDF via the print dialogue, copy, and text (`..._base_plate_design_calc.txt`). Four colour schemes are offered; the report re-reads the live design when drawn. Sections: Design forces; Column, base plate and foundation data; Design forces in the T-stubs;

Compression T-stub and foundation bearing (EN 1993-1-8 6.2.5); Tension T-stub and base plate bending (6.2.6); Bolt group resistance; Moment resistance (Table 6.7); Weld resistance (4.5.3); ETAG anchor resistance (if on); Shear transfer ($F_{f,Rd} = C_{f,d} \times N_{c,Ed}$, friction alone, shear transfer, shear to the anchors); Quantity and carbon (A1-A3, A4, A5, total with donut and bar charts). The drawing is the on-screen plan, section and actions. References listed: EN 1993-1-8:2005 (6.2 component method, Table 6.7, 4.5.3), EN 1993-1-1 Table 3.1, EN 1992-1-1 concrete strength basis, SCI P398, ETAG 001 Annex C / EOTA TR029.

5.7 Connections

Wire	Direction	What moves
Column -> Base plate	in	Section and grade; N_{Ed}, $N_{Ed,min}$, V_{Ed}, M_{Ed}. The wire has a menu: cases (DL, LL, WL, self-weight SW), shear source (V_x, V_y or resultant), moment source (M_x, M_y or resultant), and manual add-ons for N, V, M. Defaults: P, V_x, M_x, DL, LL, WL and SW on; V_y, M_y and resultants off
Foundation -> Base plate	in	Concrete class and the foundation's length, width and thickness (from the pad, cap or raft)
Base plate -> Foundation	out	Plate size (X, Y, T), column section text, base plate utilisation and carbon
Steel sections (Blue Book) -> Base plate	wire allowed	The map allows the wire, but the base plate takes its section from the Column wire; set the section on the Column
Wind designer, Sketch-ET -> Base plate	wire allowed	Check the wire menu on the map for what, if anything, it carries

How the column loads are turned into base-plate actions:

- With the case grid on: if the column rows are characteristic, the (6.10) combination is applied, and the column self-weight is added $\times$ **1.35** as a permanent action. If the rows are design values, DL + LL + WL are just summed and the self-weight is added unfactored. The least-axial combination is carried alongside for the tension side.
- With the case grid off: the column's N_{Ed}, shear and moment are used, with self-weight added.
- N_{Ed} is then clamped at zero. The source for shear and moment (V_x/V_y/resultant, M_x/M_y/resultant) fills V_{Ed} and M_{Ed}.

While a column wire is active the base plate keeps a copy of your manual N_{Ed}/V_{Ed}/M_{Ed}, column section and grade so "Restore manual" can bring them back. Turning "Use column node data" off stops the wire writing.

5.8 Tips and caveats

- Run with "Include friction check" off only when you want the anchors to take all the shear; set $C_{f,d}$ to 0 for the same effect.
- The base plate does not design the foundation. Make sure the foundation thickness and class are those you actually mean (or link them).
- Concrete properties use α_{cc} 0.85 and γ_c 1.5 and are not editable here.
- The weld check assumes $\beta_w = 0.85$ (a fixed value) and compares against the lower of column and plate f_u. The weld lengths are the standard perimeter lengths described above; any local detailing (stiffeners, shear key) is outside the tool.
- ETAG design supports at most three anchors per side. Splitting resistance is not limited (input fields exist for the data you have from the ETA, but the calculation sets splitting resistance unlimited).
- Cracked concrete is the default; select Non-cracked only where that is justified.
- If your inputs match a particular benchmark case (350 kN, 75 kN, 600 × 600 × 40 plate and so on) the panel may show a "Benchmark comparison mismatch" notice and offer a button that fills the benchmark inputs. Ignore it for real designs.
- Treat the checks as a design aid; verify the anchor product data against its European Technical Assessment.

5.9 What changed in v1.2.0

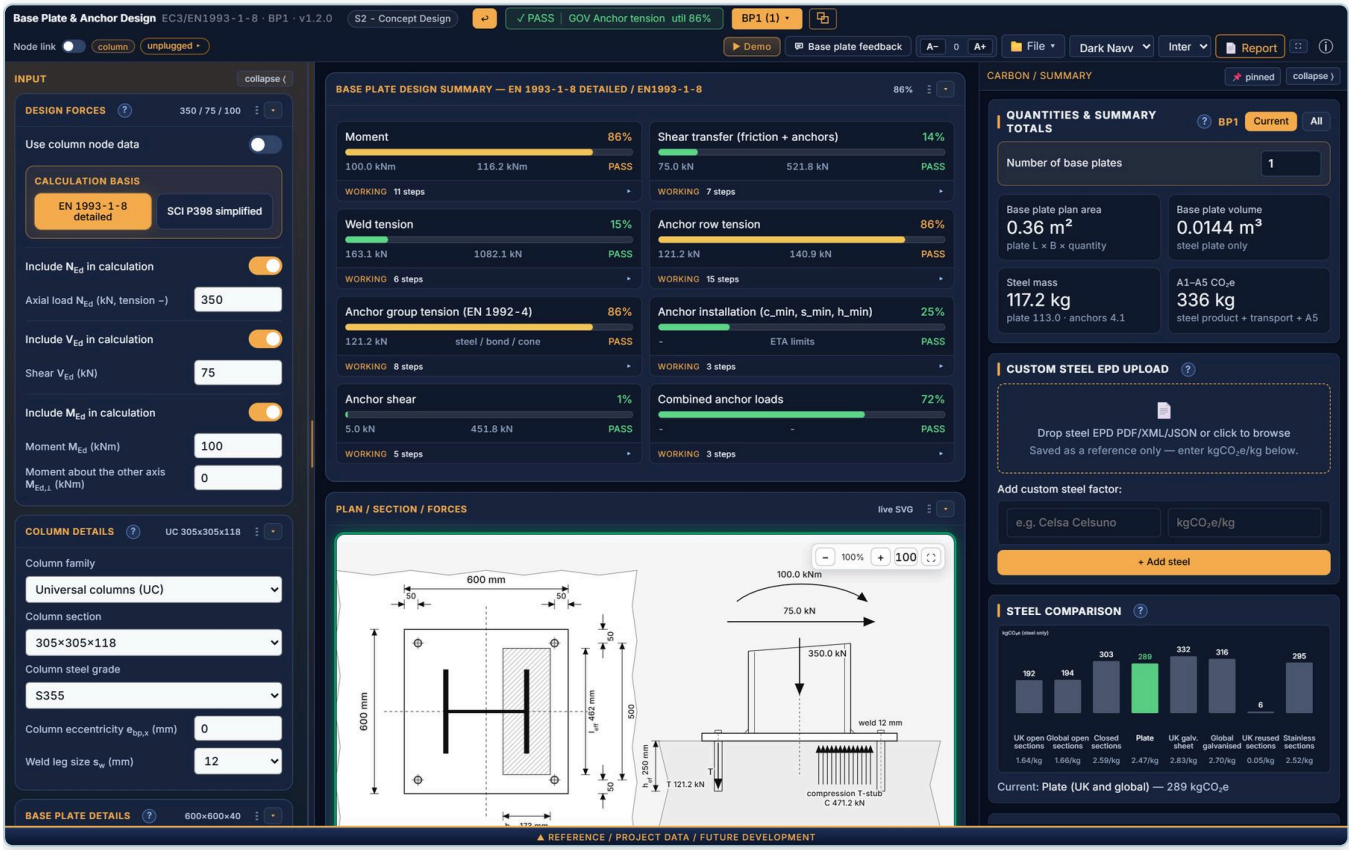
- Calculation styles renamed to "EN 1993-1-8 detailed" and "SCI P398 simplified" (third-party references removed).
- Friction now uses the column compression $N_{c,Ed}$ per EN 1993-1-8 6.2.2(6), and the shear check is friction plus anchors per 6.2.2(7).
- Steel shear of the anchor $V_{Rk,s} = k_6 A_s f_{uk}$, with $k_6 = 0.5$ for $f_{uk} > 500$ N/mm² (EN 1992-4).
- ETAG anchor card layout, toggle spacing, subscripted field labels, "Beta" label.
- Reports: subscripts kept, one font, base plate drawing label fixes.

> **References.** The standards, guidance and data sources used in this chapter are listed, with web links, in Chapter 17.

Screen reference - panel by panel

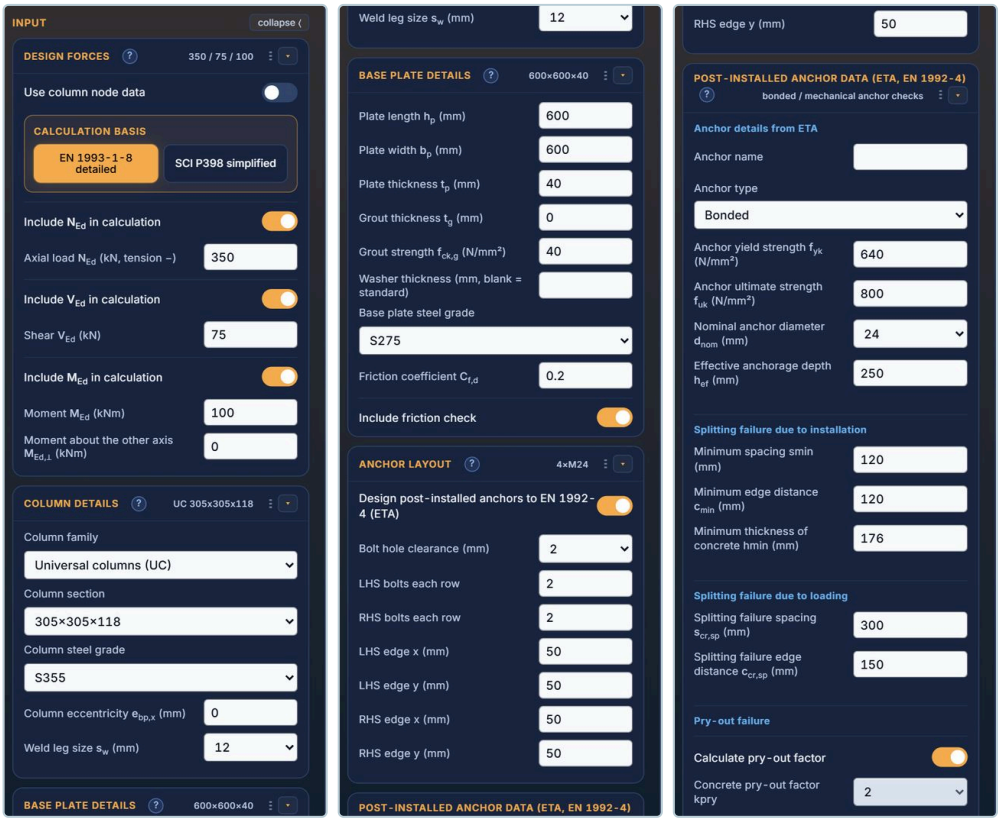
These screenshots show every panel of the module exactly as it appears on screen, scrolled from top to bottom, so you can match any field, button or result to the explanations in this chapter. Cards that are collapsed by default are shown expanded.

Base plate: the whole screen

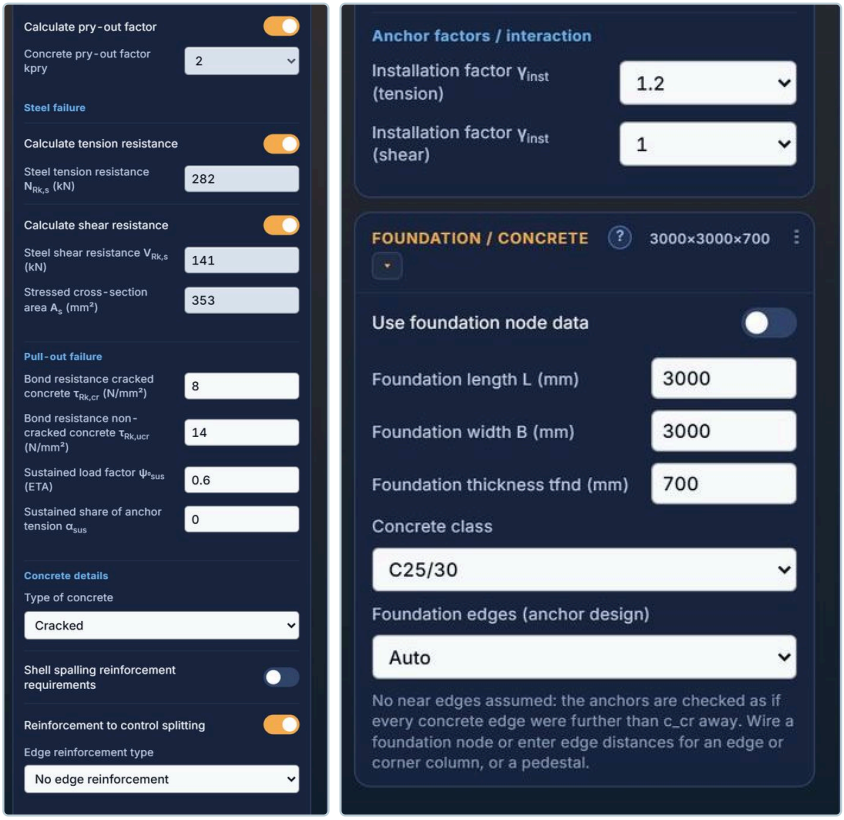


Base plate: the whole screen.

Base plate: input panel (left of the screen)

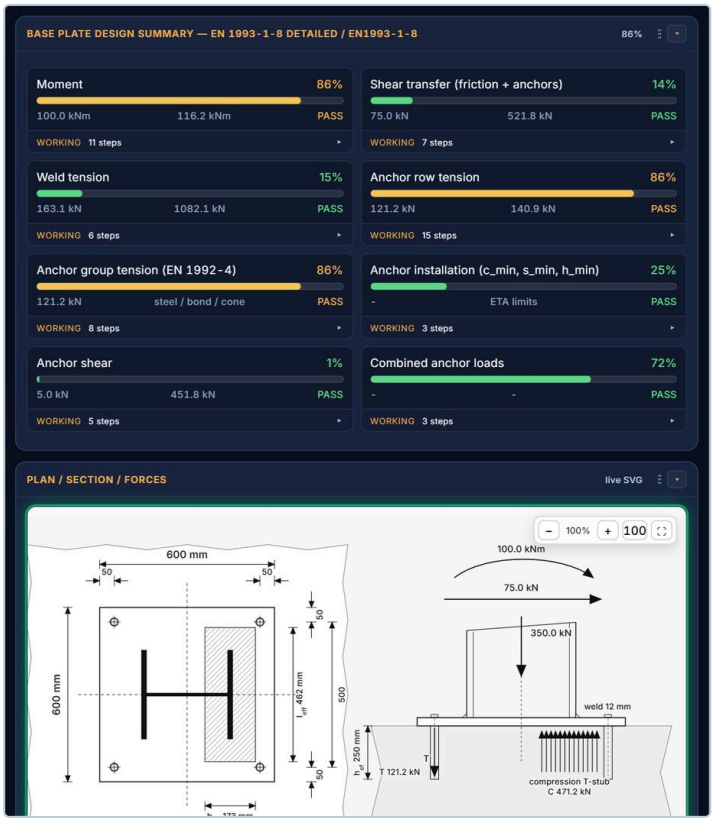


Base plate, input panel (left of the screen), parts 1-3 of 5. Cards shown: Design forces, Calculation basis, Column details, Base plate details, Anchor layout.

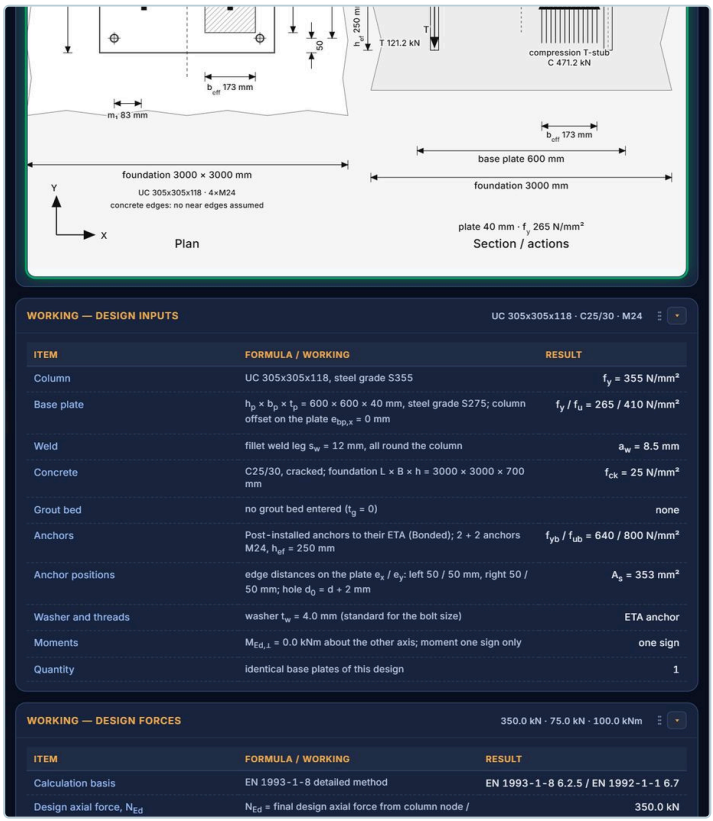


Base plate, input panel (left of the screen), parts 4-5 of 5. Cards shown: Foundation / concrete.

Base plate: working panel (centre of the screen)



Base plate, working panel (centre of the screen), part 1 of 3. Cards shown: Working.

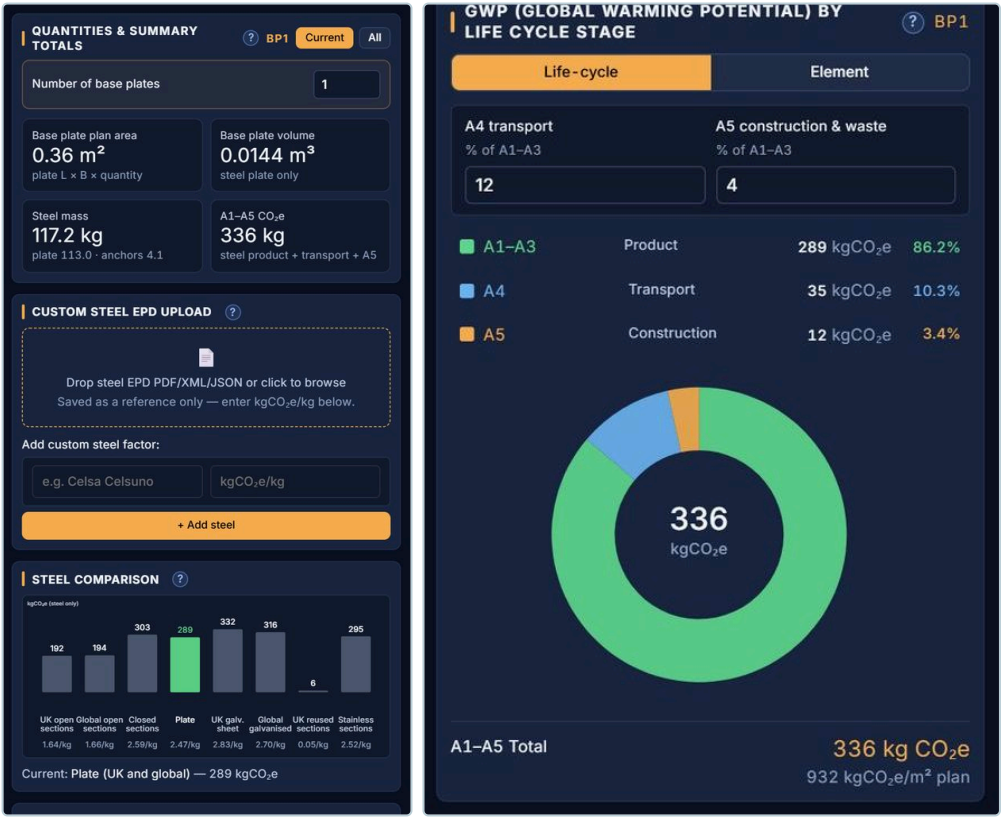


Base plate, working panel (centre of the screen), part 2 of 3. Cards shown: Working — design inputs, Working — design forces.



Base plate, working panel (centre of the screen), part 3 of 3. Cards shown: Working — design forces, Working — column / base plate / foundation data, Working — compression t-stub / foundation bearing, Working — bolt group resistance, Working — steel quantity and carbon.

Base plate: results panel (right of the screen)



Base plate, results panel (right of the screen), parts 1-2 of 2. Cards shown: Custom steel EPD upload, Steel comparison, GWP (global warming potential) by life cycle stage BP1.

CHAPTER 6

Beam analysis and design

The Beam node is a continuous-beam analyser with member design attached. You draw a beam of any length, drop supports and loads on it, and Civil ET solves it with a direct-stiffness finite-element model. It then draws shear force, bending moment and deflection live and designs the member as one of two materials: reinforced concrete to EN 1992-1-1 (EC2) or a rolled steel I-section to EN 1993-1-1 (EC3), both with the UK National Annex. A timber beam can be analysed but not designed. Every beam also gets an embodied-carbon take-off, and the whole design prints as a calculation note in the same format as the other modules.

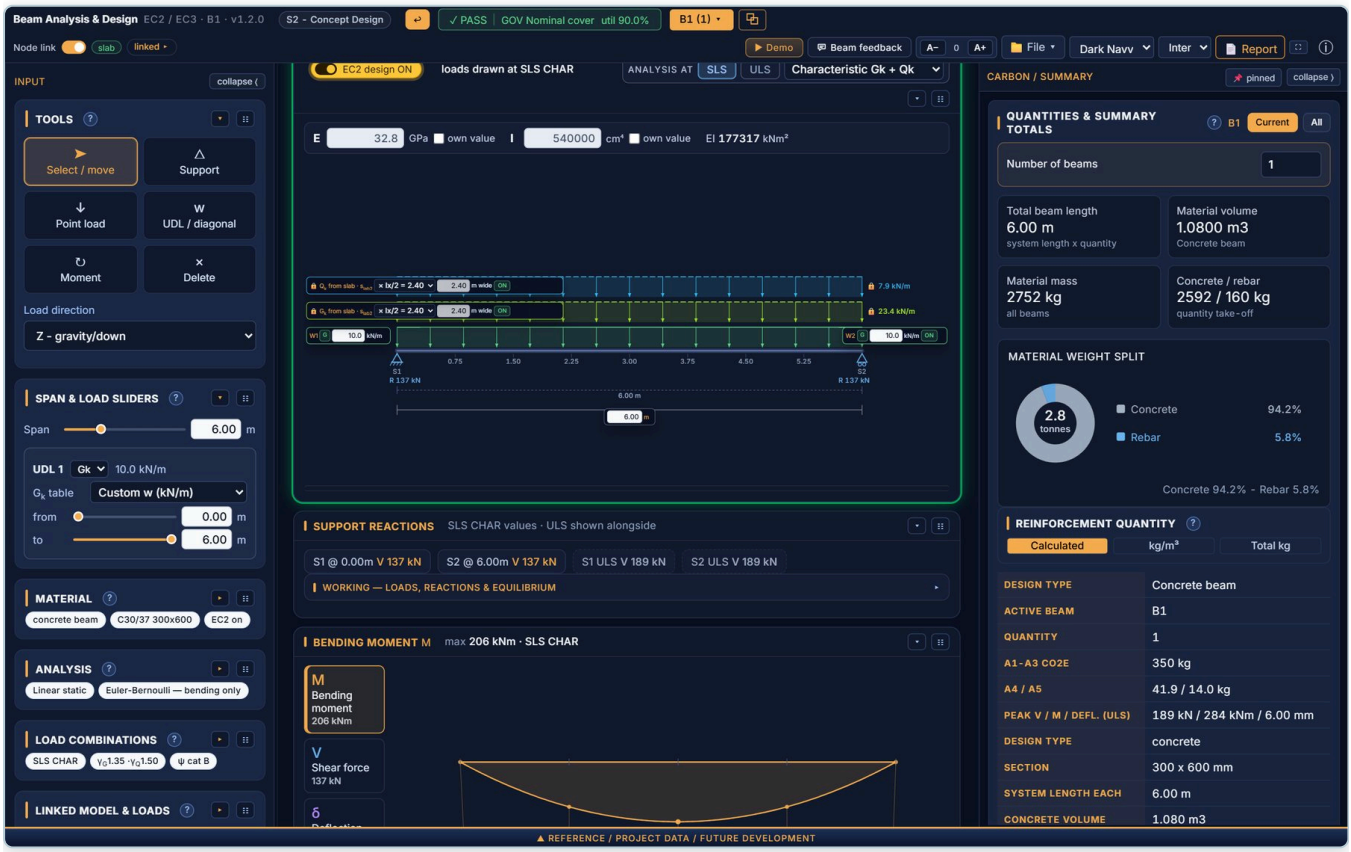
This chapter describes what the node does as built in v1.2.0. Clause numbers quoted are those used in the calculations and reports.

6.1 Opening and adding a beam

- Add **Beam analysis** from the node library (tag "M · V · δ") by clicking it or dragging it onto the map.
- **Double-click** the card (not on a button, box or drop-down) to open the full Beam workspace.
- One Beam node holds a list of designs named B1, B2, B3 and so on. The picker in the module header adds, renames, switches and deletes designs. Each design has its own state, its own output dot on the node card, and its own report. A wire leaving the card names the design it comes from.
- The workspace has three columns:
  - **Left**: the input panel, made of collapsible, re-orderable cards (Tools, Span and load sliders, Analysis, Load combinations, Material, member and design cards). The panel can be folded to a rail and resized.
  - **Centre**: the member view (supports, loads, dimensions) with the V, M and deflection diagrams and the working calculations beneath.
  - **Right**: the quantities and embodied-carbon panel.
- Global **Ctrl+Z / Ctrl+Y** undo and redo cover the whole beam design, not only the drawn geometry. **Esc** always hands the pointer back to Select / move.
- A header status chip and the node card show a PASS / FAIL verdict and the governing utilisation with the name of the governing check.

6.2 The structural model

6.2.1 Member, supports and loads



Screenshot: The beam module: tools and span sliders on the left, bending moment and shear diagrams in the middle, and support reactions below the diagrams.

Item	Meaning	Unit	Default
Span L	Developed length of the member	m	6 (limits 0.1 to 999)
Supports	Each has a position x and a type: **pinned**, **roller** or **fixed**	m	pinned at 0, roller at L
Point load	P at x	kN	-
UDL	w_1 to w_2 (trapezoidal if they differ) between x_1 and x_2	kN/m	one 10 kN/m permanent UDL over the full span
Moment	M at x	kNm	-
Class	Each load is **G** (permanent) or **Q** (variable)	-	G
Load direction	Z (gravity, down) or Y (up/down in the view)	-	Z

Notes on supports:

- Pinned and roller are the same in this planar model: vertical displacement restrained, rotation free. Fixed restrains vertical displacement and rotation. There is no axial degree of freedom.
- Any number of supports is allowed. The node refuses to solve a mechanism and says so ("Beam is a mechanism - add another support to restrain rotation"). It needs two supports at different positions, or one fixed support.
- Overhangs beyond the outer supports and cantilevers (a single fixed support) are handled.

Drawing tools (Tools card): **Select / move**, **Support**, **Point load**, **UDL / diagonal**, **Moment**, **Delete**. Pick a tool and click the member. Supports and loads can be dragged, and every position and magnitude also has a slider and a typed box in the "Span and load sliders" card. Support positions snap to a grid (0, 0.25 or 0.5 m, default 0.25). The member can be drawn at an angle (angle snap $0°$, $15°$, $30°$, $45°$ or $90°$); the load direction is then projected onto the member normal, which is oriented so that gravity always loads the beam downward whichever way it was drawn.

UDLs can be filled from libraries: the **Gk table** (dead loads, CCIP-041 Table 2.14) and the **Qk table** (EN 1991-1-1 imposed categories, CCIP-041 Tables 2.2 to 2.8). Pick an entry and give a loaded width, and the UDL becomes area load x width. A loaded width is never invented silently; the box starts at 1.00 m.

Loads that arrive by wire (6.9) are drawn as locked bands and cannot be edited by hand. Each band can be switched off ("muted"), which keeps it on screen but leaves it out of every solve.

6.2.2 Stiffness: E and I

The analysis stiffness EI comes from the material, and can be overridden by ticking "own value" on the member strip beside the drawing.

Material	E	I
Concrete	$E_{cm} = 22\,[(f_{ck}+8)/10]^{0.3}$ GPa (EC2 Table 3.1), 32.8 GPa for C30/37	$bh^3/12$ of the rectangular beam (default 300 × 500 mm gives 312 500 cm^4)
Steel	210 GPa (EN 1993-1-1 3.2.6)	I_y (or I_z for minor-axis bending) of the selected Blue Book section
Timber	11 GPa (editable concept value)	$bh^3/12$

Internally EI is $E\,[\mathrm{GPa}] \times I\,[\mathrm{cm}^4]/100$ in kNm^2. The concrete stiffness is uncracked and elastic; cracking is handled in the long-term deflection check (6.5.6), not in the diagrams.

6.2.3 Bending axis

Every material can be designed about the **major** axis (default) or the **minor** axis (member turned on its side). The switch sits on the drawing. It changes which inertia drives the analysis and which section modulus, shear area and width the design uses. About the minor axis a steel beam cannot buckle laterally, the flanges carry the shear, and the web does not classify. For concrete and timber the width and depth simply swap.

6.3 Analysis modes

The Analysis card lists five options:

Option	Status in v1.2.0
Linear static	Always on. Direct-stiffness solver.
Euler-Bernoulli	Always on. Bending-only beam element.
Timoshenko / shear	**Available, off by default.** Tick it to add shear flexibility.
Non-linear	Shown but locked: "separate future check - not implemented".
Vibration	Shown but locked: "separate future check - not implemented".

Do not rely on a non-linear or vibration result from this node; there is none.

Timoshenko. With the option on, each element uses the full Timoshenko stiffness with $\phi = 12EI/(GA_sL_e^2)$, so the slope becomes $\theta + V/(GA_s)$ and deflections grow while the statics are untouched. GA_s is:

- steel: G = 81 000 N/mm^2 times the EN 1993-1-1 6.2.6(3) shear area A_v (the flange shear area for minor-axis bending);
- concrete: $G = E_{cm}/2.4$ ($\nu = 0.2$, EC2 3.1.3(4)) times $5/6\,bh$;

- timber: $G = E/16$ times $5/6\,bh$.

The effect scales with $(\text{depth}/\text{span})^2$, about 3 % on a 533 UB at 8 m, but 10 % to 25 % on short deep members. The fixed-end load vectors stay the Euler-Bernoulli ones. That is exact for a load symmetric on its member and for any determinate beam, but a member fixed at both ends under an unsymmetric load keeps its Euler-Bernoulli end moments. Leave the option off to match a hand calculation, or on to line up with frame packages that include shear deformation.

The solver samples extra stations either side of every support and load so that jumps in shear and moment are drawn and checked correctly.

6.4 Load combinations (EN 1990)

6.4.1 Loading mode

The **Loading mode** (Load combinations card) is one of:

- **Characteristic G_k / Q_k - EN 1990 combinations** (default for new beams). Each load is classed G or Q and factored by the node.
- **Pre-factored design loads ($\gamma = 1.0$)**. A legacy mode kept so older saved beams open unchanged. The entered loads are used as typed for both ULS and SLS, and there is no separate quasi-permanent set. Self-weight still takes 1.35 at ULS.

6.4.2 ULS expression

Setting	Result
$\gamma_G = 1.35$ (default)	Exp. (6.10): $n = 1.35G_k + 1.50Q_k$
$\gamma_G = 1.25$ ("6.10a / 6.10b - the less favourable")	The code evaluates both on the factored totals and uses the worse. Exp. (6.10a) = $1.35G_k + 1.5\psi_0 Q_k$, Exp. (6.10b) = $1.25G_k + 1.5Q_k$. The working and report state which one governed

6.4.3 ψ factors and SLS combinations

The ψ **load category** (EN 1990 Table NA.A1.1, same table as the slab) sets ψ_0, ψ_1, ψ_2:

Category	ψ_0	ψ_1	ψ_2
A residential, B office (default B)	0.7	0.5	0.3
C congregation, D shopping	0.7	0.7	0.6
E storage	1.0	0.9	0.8
F traffic, vehicle up to 30 kN	0.7	0.7	0.6
G traffic, 30 to 160 kN	0.7	0.5	0.3
H roofs	0.7	0	0

Service combinations (applied to Q loads; G loads are always 1.0):

SLS combination	Expression
Characteristic	$G_k + Q_k$
Frequent	$G_k + \psi_1 Q_k$
Quasi-permanent	$G_k + \psi_2 Q_k$

6.4.4 What the diagrams show, and what is designed

The **Analysis read at** switch (member view header and Load combinations card) chooses what the reactions and the V / M / δ diagrams are drawn at: **SLS** (default, with a choice of the three SLS combinations) or **ULS**. It changes only what you read. The design checks do not follow it:

- Strength checks (flexure, shear, LTB and so on) always run on the ULS combination.
- Deflection and crack control always run on the quasi-permanent combination for concrete.
- Steel deflection runs on unfactored variable actions and on the total, both on characteristic loads (6.6.5).

6.4.5 Self-weight

The **Member self-weight included** switch (on for a new beam) adds the beam's own weight as one permanent line load over the full length:

- Concrete (EC2 design on): $b \times h \times 25$ kN/m³ (EN 1991-1-1 Table A.1). The default 300 × 500 beam adds 3.75 kN/m.
- Steel (EC3 design on): Blue Book mass per metre $\times\, 9.81/1000$.
- Timber, or design switched off: nothing is added.

The superimposed loads you enter exclude self-weight. Switch it off if you have already entered it. Beams saved before this switch existed do not get it automatically.

6.5 Concrete beam design (EC2)

Choose **Concrete beam** in the Material card. The design switch "EC2 concrete design" is on by default; switching it off leaves an analysis-only beam (no checks and no quantities).

6.5.1 Inputs

Input	Unit	Default
Width $b\times$ depth h	mm	300 × 500
Concrete grade (C20/25, C25/30, C28/35, C30/37, C32/40, C35/45, C40/50, C50/60 or custom)	-	C30/37 (f_{ck} 30)
f_{yk}	N/mm²	500
f_{ctm}	N/mm²	EC2 Table 3.1 value for the grade (typed only for a custom grade)
Nominal cover to the links	mm	30 ("auto" cover mode takes the computed minimum)
Link diameter	mm	8
Bottom bars ($n\times$ diameter)	-	3 × 20
Top bars ($n\times$ diameter)	-	3 × 25
Shear links: legs, diameter, spacing	-, mm, mm	2, 8, 250
Aggregate size	mm	20
Redistribution ratio δ (0.7 to 1.0)	-	1.0
Ductility cap K' of 0.168	on/off	off
Fire period (R)	min	60 (0 = not checked)
Monolithic end supports	on/off	on
Exposure class	-	XC1, dry / permanently wet
Design working life	years	50 (or 100)

Input	Unit	Default
Special QC of the concrete production	on/off	off
Cover mode	auto / user	auto for new beams
Δc_{dev}	mm	10 (or 5 or 0)
Cast against	formwork / blinding / soil	formwork
$c_{min,dur}$ override	mm	0 (automatic)
Support width	mm	300
Support kind	direct / indirect	direct
Bar end at the support	straight / bend / continuous	bend
Curtailed bottom bars: number, distance from the support	-, mm	0 (all bars run full length), 1000
Curtailed top bars: number, distance from the support	-, mm	0, 1500
Crack k basis	EC2 / conservative	EC2

All design data are per face: "bottom" and "top" bars are the member's sides about the minor axis.

6.5.2 Flexure (EC2 6.1, 3.1.7, 5.5)

Run separately for the sagging (bottom) and hogging (top) faces, with M_{Ed} taken from the solved ULS envelope.

- $d = h - c_{nom} - \phi_{link} - \phi/2$. The opposite-face bar centre $d_2 = c_{nom} + \phi_{link} + \phi/2$.
- $\gamma_C = 1.5$, $\alpha_{cc} = 0.85$, $\gamma_S = 1.15$; $f_{yd} = f_{yk}/1.15$.
- Rectangular stress block with $\lambda = 0.8$ and $\eta = 1.0$ up to C50/60; for higher classes the EC2 3.1.7(3) Exp. (3.19) to (3.22) values are used, with ε_{cu2} from Table 3.1. The UK NA values $k_1 = 0.4$ and $k_2 = 0.6 + 0.0014/\varepsilon_{cu2}$ set the neutral-axis limit (5.5(4)).
- $K = M_{Ed}/(bd^2 f_{ck})$. K' follows from δ and the stress block, optionally capped at 0.168.
- K up to K': singly reinforced, $z = 0.5d\left[1 + \sqrt{1 - 2K/(\eta\alpha_{cc}/\gamma_C)}\right]$ limited to $0.95d$; $A_s = M_{Ed}/(f_{yd} z)$.
- K above K': **doubly reinforced**. The balanced moment is taken at K', the excess is resisted by compression steel $A_{s2} = \text{excess}/\left(f_{yd}(d - d_2)\right)$. The compression steel must actually be provided on the opposite face. The node compares A_{s2} with the bars there and fails the beam if they are short.
- $A_{s,min} = \max\left(0.26\dfrac{f_{ctm}}{f_{yk}}\,bd,\ 0.0013\,bd\right)$ (9.2.1.1(1)), applied only to a face where the moment puts it in tension. $A_{s,max} = 0.04A_c$, tested against the area **provided** (9.2.1.1(3)) as well as required.
- Clear bar spacing must be at least $\max(\text{bar diameter},\ \text{aggregate} + 5\ \text{mm},\ 20\ \text{mm})$.
- The moment resistance M_{Rd} of the bars provided is worked out (stress block, with the opposite face counted as compression steel above the ductility limit), so the reported flexural utilisation is M_{Ed}/M_{Rd}, together with the $A_{s,min}$ ratio. Where compression steel is needed the ratio is $A_{s,req}/A_{s,prov}$.

Monolithic end supports (9.2.1.2(1)). With the switch on (default), an end support modelled as pinned or roller still attracts hogging: the top face is designed for the larger of the analysed hogging moment and $\beta_1\times$ the adjacent span moment, with $\beta_1 = 0.25$ (UK NA; EC2 recommends 0.15). Interior supports of a continuous beam and fixed supports already carry their moment from the analysis. Switch it off for a beam sitting on bearings, steelwork or masonry.

6.5.3 Shear (EC2 6.2)

The design shear V_{Ed} is the peak absolute shear on the ULS diagram (no reduction for load near the support is applied here).

1. **Without links**: $V_{Rd,c} = \left[\frac{0.18}{\gamma_C}\, k\, (100\rho_l f_{ck,v})^{1/3}\right] bd$ with a floor of $v_{min} = 0.035k^{1.5}\sqrt{f_{ck,v}}$, $k = 1 + \sqrt{200/d}$ capped at 2.0. ρ_l uses the tension steel on the face that is in tension at the shear-critical section (read from the moment within d of the peak shear), capped at 0.02. If bottom bars are curtailed (below) only the bars running through are counted. UK NA 3.1.2(2)P: for classes above C50/60, $f_{ck,v}$ is taken as 50.
2. **Strut and links (6.2.3)**: $\nu_1 = 0.6(1 - f_{ck,v}/250)$; $f_{cd} = f_{ck,v}/1.5$ ($\alpha_{cc} = 1.0$ for shear, UK NA 3.1.6(1)P); $z = 0.9d$. The strut angle is **variable** within $1 \le \cot\theta \le 2.5$ (UK NA). The node starts at $\cot\theta = 2.5$ ($21.8°$) and steepens θ only if $v_{Ed,z}$ exceeds $v_{Rd,max}$ at 2.5, using $\theta = 0.5 \arcsin\left[2v_{Ed,z}/(\nu_1 f_{cd})\right]$. If V_{Ed} exceeds the capacity even at $45°$ the section or the concrete class must increase.
3. **Links**: $A_{sw}/s = V_{Ed}/(z f_{ywd} \cot\theta)$ when V_{Ed} exceeds $V_{Rd,c}$, never less than $A_{sw,min} = 0.08b\sqrt{f_{ck}}/f_{yk}$ per mm. The provided links (legs x area / spacing) must meet that, and the spacing must not exceed $0.75d$.
4. The strut utilisation is $V_{Ed}/V_{Rd,max}(\theta)$; the $45°$ capacity is kept as the hard limit.

6.5.4 Crack control (EC2 7.3)

- **7.3.2 minimum steel**: $A_{s,min} = k_c k f_{ct,eff} A_{ct}/\sigma_s$ with $k_c = 0.4$, k from 1.0 (h up to 300 mm) to 0.65 (h of 800 mm or more), $f_{ct,eff} = f_{ctm}$, A_{ct} the tension zone of the uncracked section. σ_s is the larger of the stresses allowed by Tables 7.2N and 7.3N for $w_k = 0.3$ mm that the bars provided satisfy, capped at f_{yk}. The "conservative" basis takes k from $\min(h, b)$ and Table 7.3N only.
- **7.3.3 without direct calculation**, on **every** face in tension (including the top face over supports and under the 9.2.1.2 moment): the steel stress is estimated from $f_{yd} \times (A_{s,flex}/A_{s,prov}) \times (\text{quasi-permanent}/\text{ULS moment})/\delta$, and bar size (Table 7.2N, corrected by Exp. (7.6N)) or bar-centre spacing (Table 7.3N) is checked against $w_{max} = 0.3$ mm (UK NA Table NA.4 for reinforced members under the quasi-permanent combination).
- Beams 1000 mm deep or more get a reported skin-reinforcement area (7.3.3(3), $k = 0.5$, $\sigma_s = f_{yk}$). Side bars are reported, not designed.

6.5.5 Cover, fire, anchorage and curtailment

Nominal cover (EC2 4.4.1, UK NA, BS 8500-1 Table A.4). $c_{nom} = c_{min} + \Delta c_{dev}$, with $c_{min} = \max(c_{min,b};\ c_{min,dur};\ 10\ \text{mm})$. $c_{min,b}$ is the bar diameter (plus 5 mm for aggregate over 32 mm). $c_{min,dur}$ is the larger of the BS 8500-1 Table A.4 value for the exposure class and concrete strength, and the EC2 Table 4.4N value at the Table 4.3N structural class for the working life and QC choice. The Δc_{dur} terms are zero (UK NA). Concrete cast against blinding needs at least 40 mm, against soil 75 mm. The entered cover is to the **links**; the main bars sit a link diameter further in and are checked separately.

The status has three states:

- **OK**.
- **Review required**: the bond minimum is met but the durability cover for the chosen exposure class, or the concrete class required by BS 8500-1, is not. The exposure class is the designer's decision, so this does not fail the beam.
- **FAIL**: bond minimum or casting-against-ground cover not met.

In **auto** cover mode a new beam takes the computed minimum (rounded up to 5 mm), so it passes by construction and ranks near 100 % without being the real governing check. Beams saved earlier keep their own cover in "user" mode.

Fire (EN 1992-1-2). With a fire period of 30, 60, 90, 120, 180 or 240 minutes, the beam is checked against Table 5.5 (simply supported) or Table 5.6 (continuous: more than two supports, or any fixed support): minimum width b_{min} and axis distance a. The axis distance is $c_{nom} + \phi_{link} + \phi/2$, plus 10 mm in the required axis distance where b is below the widest tabulated width (corner bars). Other fire periods are not offered; 0 switches the check off.

Anchorage and laps (EC2 8.4, 8.7.3). For the bottom and top bars: $f_{bd} = 2.25\eta_1\eta_2 f_{ctd}$ ($f_{ctd} = 0.7 f_{ctm}/1.5$, $\alpha_{ct} = 1.0$ UK NA), $l_{b,rqd} = (\phi/4)(\sigma_{sd}/f_{bd})$, $l_{bd} = \alpha$ factors $\times\, l_{b,rqd}$ with minimum $\max(0.3 l_{b,rqd};\ 10\phi;\ 100\ \text{mm})$, and lap length l_0 with α_6 from the share of bars lapped (100 % gives 1.5) and minimum $\max(0.3\alpha_6 l_{b,rqd};\ 15\phi;\ 200\ \text{mm})$. Bond condition (good/poor) follows Figure 8.2. Both the length at the actual stress and at full f_{yd} are reported. α_3, α_4 and α_5 are taken as 1.0 in the general table (safe side).

End anchorage and curtailment (9.2.1.3 to 9.2.1.5).

- At each end support (when the beam end lies inside the support) the force to anchor, F_E, comes from the shear and the shifted moment diagram. Required l_{bd} for the bar end you chose (straight, 90-degree bend, or continuous through) is compared with the length available in the support width. For a direct support a transverse-pressure factor is allowed ($\alpha_5 = 1 - 0.04p$, within 0.7 to 1.0).
- At least $\beta_2 = 0.25$ of the span bottom steel must reach the supports (9.2.1.4(1)).
- Curtailed bars: the tensile force shifted by $a_l = z\cot\theta/2$ is compared with the force the bars that continue can develop, beyond the curtailment point plus anchorage (Figure 9.2). The bars left after curtailment must still meet $A_{s,min}$.
- With nothing curtailed (default) every bar runs the full length.

6.5.6 Serviceability (EC2 7.4)

Run on the quasi-permanent solve.

1. **Span / effective depth (7.4.2, Exp. 7.16a/b)**: allowable $l/d = N \times K \times F_1 \times F_2 \times F_3$, capped at $40K$ (UK NA Table NA.5 Note 6).
   - N from $\rho = A_{s,req}/(bd)$ against $\rho_0 = \sqrt{f_{ck}}/1000$; compression steel ignored (conservative).
   - K from Table 7.4N: simply supported 1.0, end span of a continuous beam 1.3, interior span 1.5, cantilever 0.4. The default is automatic from the supports (a two-support beam with one fixed support reads 1.3, with both 1.5); you can force a value.
   - $F_1 = 1.0$ (rectangular section); $F_2 = 7/\text{span}$ only when "Brittle partitions" is ticked and span exceeds 7 m; $F_3 = \min(1.5,\ A_{s,prov}/A_{s,req})$.
   - The governing span is the longest span between supports; the tension face is the one with the larger $A_{s,req}$.
2. **Explicit long-term deflection**: the quasi-permanent elastic deflection is scaled by $EI_{elastic}/EI_{eff}$. Creep is taken as $\varphi = 2.0$ (fixed in the beam node); $E_{eff} = E_{cm}/(1+\varphi)$; cracking decided from $M_{cr} = 0.9 f_{ctm} I_u/(h - x_u)$ against the quasi-permanent moment; with $\zeta = 1 - 0.5(M_{cr}/M)^2$ the curvatures of the cracked and uncracked states are interpolated (EC2 7.4.3 approach, following CCIP-060). The check is against **span/250**. Span/500 is also calculated and shown.
3. **Crack control** as in 6.5.4.

The verdict includes span/depth, the span/250 deflection and crack control as well as every strength check.

6.5.7 Concrete outputs and governing utilisation

The governing utilisation ranks these named checks: Flexure (bottom), Flexure (top), $A_{s,min}$ crack, Fire, Nominal cover, End anchorage, Curtailment, Compression steel (A_{s2}), Shear links, Shear strut, Deflection, Span/depth, Crack control. The beam passes only if every one passes. All warnings are listed in plain words (for example "Bottom reinforcement is below the positive moment As required").

An **optimiser** ("Find optimised beam options") searches nearby widths and depths with bar and link choices and lists the passing designs ranked by A1-A3 embodied carbon. Nothing is searched until you press the button; the results stay until an input changes, and then the button returns with an "inputs changed" note.

6.6 Steel beam design (EC3)

Choose **Steel beam**. The "EC3 steel beam design" switch is on by default. The workflow follows SCI P364 (Steel Building Design: Worked Examples, Open Sections): restrained beam (Ex 2), unrestrained beam (Ex 3), beam with intermediate restraints (Ex 4) and continuous beam (Ex 6). Resistances use the embedded Steel for Life Blue Book section data and are cross-checked against the published Blue Book tables.

6.6.1 Inputs

Input	Options	Default
Section family	Universal beams (UB), Universal columns (UC)	UB
Section	any Blue Book section	356x171x45
Steel grade	S275, S355, S460	S355
Bending axis	major (y-y), minor (z-z)	major
Lateral restraint	Fully restrained (no LTB); supports only; supports + point-load positions; custom positions	fully restrained
Custom restraint positions	list of x (m)	empty
Restrained flange	both; top only; bottom only	both
C_1	auto (fitted to the moment diagram) or a Blue Book row value (1.00, 1.13, 1.35, 1.50, 1.77, 2.00, 2.50) or a custom value (1.0 to 2.8)	auto
Support / load introduction	connection; bearing on the bottom flange	connection
Stiff bearing length s_s	mm	50
Deflection limit on variable actions	auto, 360, 250, 200, 180	auto
Total ($G_k + Q_k$) deflection limit	200, 250, 360, off	200

Selecting a section loads its properties and sets the analysis I from the Blue Book. The node warns if the analysis E or I differs from the section (for example a custom inertia, or E not 210 GPa).

6.6.2 Checks

Material: $\gamma_{M0} = \gamma_{M1} = 1.0$ (UK NA NA.2.15). Yield strength f_y is thickness-stepped from the controlling element (the larger of flange and web thickness) with the EN 10025 grade properties. Shear modulus G = 81 000 N/mm²; E = 210 000 N/mm².

1. **Classification (EN 1993-1-1 Table 5.2)** for pure bending, using $\varepsilon = \sqrt{235/f_y}$. Flange outstand limits 9ε, 10ε, 14ε; web limits 72ε, 83ε, 124ε. Class 1 and 2 use the plastic modulus, class 3 the elastic modulus. About the minor axis the whole outstand is in compression, and the web does not classify.
   - **Class 4 is flagged, not designed**: no effective-section design is implemented. The beam is reported as failing the "Section class 4 (slender)" check.
2. **Shear (6.2.6)**: $V_{pl,Rd} = A_v(f_y/\sqrt{3})/\gamma_{M0}$ with $A_v = A - 2bt_f + (t_w + 2r)t_f$, not less than $\eta h_w t_w$ ($\eta = 1.0$, UK NA). For minor-axis bending $A_v = A - \sum(h_w t_w)$. If h_w/t_w exceeds $72\varepsilon/\eta$, shear buckling to EN 1993-1-5 section 5 must be checked separately; the node flags this as failing.
3. **Bending (6.2.5) with moment-shear interaction (6.2.8)**: $M_{c,Rd} = W_{pl}$ (or W_{el}) f_y/γ_{M0}. The interaction is checked at every sampled station, so the one-sided shear next to an internal support is captured. Where V is above $0.5V_{pl,Rd}$ the resistance is reduced with $\rho = (2V/V_{pl,Rd} - 1)^2$ (capped at 1.0, using 6.2.8(5) for major axis and 6.2.8(3) for minor).

4. **Lateral-torsional buckling (6.3.2)**, segment by segment between lateral restraints. Rolled-section method (6.3.2.3) with the UK NA parameters: $\bar{\lambda}_{LT,0} = 0.4$, $\beta = 0.75$, buckling curve by h/b (b for h/b up to 2, c up to 3.1, d beyond), and the modification factor f with $k_c = 1/\sqrt{C_1}$ (NA.2.18). M_{cr} is the doubly symmetric fork-end expression of SCI NCCI SN003b. C_1 is fitted to each segment's actual moment diagram: the straight-line expression $C_1 = 1.77 - 0.88\psi + 0.11\psi^2$ for linear diagrams, otherwise the Serna et al expression (D. Brown, NSC Nov/Dec 2013).
   - Segments run between real lateral/torsional restraints. A point of contraflexure is not a brace. Supports are taken as fork ends.
   - If restraints hold only one flange, they count only where that flange is the compression flange (top flange for sagging, bottom for hogging), so two families of segments are checked.
   - Unrestrained overhang tips use the physical overhang length with $C_1 = 1.0$; check the torsional restraint at the support yourself.
   - A fixed support is not treated as a minor-axis or torsional restraint.
   - The governing segment is reported. A negligible-moment segment is skipped.
   - Minor-axis bending cannot suffer LTB.
5. **Web bearing and buckling (EN 1993-1-5 section 6)** at supports and point loads: patch loading types (a) and (c), k_F, l_y, $\chi_F = 0.5/\bar{\lambda}_F$, with the stiff bearing length s_s limited to h_w. **The default "connection" setting reports the check but does not count it**, because a beam framed onto a fin or end plate does not load its web as a patch. Choose "bearing on the bottom flange" to count it. If the check fails while set to "connection" a warning still says so.
6. **Deflection (UK NA NA.2.23)**: every span between supports gets its own limit, and each overhang is checked as a cantilever.
   - Variable (Q_k only) deflection against span/360 for spans and L/180 for cantilevers in **auto** (brittle finishes), or against a fixed denominator you choose.
   - The **total** ($G_k + Q_k$) deflection is also checked, against span/200 by default (cantilevers take half the denominator). This second check is off in the legacy pre-factored mode, where the entered loads are already the service loads and the first check is on the total.
   - Shear deformation is included only if the Timoshenko option is on, and the report states which.

6.6.3 Results

The utilisation list ranks Shear, Web shear buckling (if tripped), Section class 4 (if tripped), Bending, LTB, Web bearing (if counted), Deflection (variable) and Deflection (total). Blue Book comparison rows show $M_{c,y,Rd}$, $M_{b,Rd}$ at the governing length and C_1, $V_{c,Rd}$ and $F_{Rd}(c = 0)$ for the same section and grade, and say so when a value is not tabulated.

6.6.4 What the steel design does not cover

Stated in the working and the warnings:

- biaxial bending (6.2.9). The node designs one axis at a time on the moments of the planar analysis;
- bending with torsion (P364 Ex 5), composite beams (Ex 7/8), sub-grade selection;
- shear buckling to EN 1993-1-5 section 5, Class 4 effective sections;
- destabilising (top-flange free) loading and tension-flange restraint credit (SCI P360);
- section classification under axial force.

The optimiser ("Find lighter steel sections") designs candidate sections in the family and lists passing ones by embodied carbon. As for concrete, it only runs on a button press.

6.7 Timber beams

Timber is **analysis only** in this release. The material list shows it locked ("design coming"). A saved timber beam still analyses: you get reactions and V / M / δ diagrams, a rectangular section $b \times h$ (defaults 100 × 300 mm), density (450 kg/m³), E (11 GPa). There are no timber design checks and no automatic self-weight. The only utilisation shown for a

timber beam (and for a concrete or steel beam whose design switch is off) is deflection against span/250 on the quasi-permanent solve. The take-off and carbon figures are produced (6.8).

6.8 Quantities and embodied carbon

The right panel gives the take-off and A1 to A5 carbon, using the IStructE "How to calculate embodied carbon" (3rd ed.) Table 2.5 recommended factors as the default data set.

Item	Default
Quantity (identical beams)	1
Steel factor	UK rolled open sections (consumption average), 1.64 $kgCO_2e/kg$
Concrete factor	CEM II/A-L, 217 $kgCO_2e/m^3$
Rebar factor	UK reinforcement bar (consumption average), 0.72 $kgCO_2e/kg$
Timber factor	0.25 $kgCO_2e/kg$ (typical structural timber)
A4 transport	12 % of A1-A3
A5 construction	4 % of A1-A3
Scope	the current beam or all beams on the node

- **Steel**: mass = Blue Book kg/m $\times$ length $\times$ quantity (volume = mass / 7850).
- **Concrete**: volume = $b \times h \times L \times$ quantity, mass at 2400 kg/m^3. **Reinforcement** with the design on: bar masses from the bottom and top bars actually provided (curtailed bars at their shortened length), plus links (count from the spacing, perimeter length from the cover and link diameter plus $20\times$ link diameter for hooks). With the design off, kg/m^3 allowance.
- **Timber**: volume x density.
- If the EC2 or EC3 design is **off**, the beam is analysis only: every quantity and carbon figure is zero and flagged "not assessed".

The charts compare factor choices (concrete cement types, steel and rebar sources, timber) and a custom factor or EPD can be added and stays selected.

Bar-bending-schedule override (concrete only). The take-off is based on straight bars, links and hooks, so it reads low compared with a real bending schedule. The reinforcement-quantity card lets you replace it with your own number, per beam before the quantity multiplier:

- **Auto**: the derived mass;
- **Rate**: kg per m^3 of concrete (thumb-rule presets by element type, or your own);
- **Total kg**: a scheduled quantity per beam.

The card shows the derived value for comparison. Beam, slab, column and foundation all use this same control.

6.9 Connections to other nodes

6.9.1 Into a beam (the IN dot)

Source	What arrives
Slab	The slab's characteristic area loads g_k and q_k (kN/m², including slab self-weight) become UDL bands: area load x loaded width. A one-way slab gives a uniform band. A two-way slab sheds at 45 degrees: a trapezoid on the long-edge beam, a triangle on the short-edge beam. Loaded width defaults to $l_x/2$ (edge beam); choose full l_x (internal beam) or type a width, per class (G_k and Q_k bands have their own controls). The wire can also set the beam span from the slab edge length. The ψ category is copied from the slab.
Loads (load set)	Line rows (kN/m) become full-span UDLs, vertical force rows (kN, F_z) become mid-span point loads (DL as G; LL and HL as Q), and area rows (kN/m²) become UDLs through a loaded width that starts at 1 m.
Math	A kN or kN/m answer is placed the same way as a load row; a kN/m² answer is treated as an area load. A plain number becomes a permanent UDL.
IFC	The designed element's span, material and section, with scavenged tributary loads and the beam's own weight.

Rules:

- Any wire that writes loads switches the beam to **characteristic** mode.
- Wired bands are locked and tagged with their source. Deleting a wire removes the bands it wrote (only that wire's, even when several wires from one source feed the beam).
- "Type my own loads instead" (Supports and loads card) keeps span, material and section coming across but stops the wires touching the loads; the bands become ordinary rows.
- A wire carrying the wrong unit is rejected, with the reason shown.
- The map also lets you draw Wind and Sketch-ET wires to a beam (the connection is permitted), but the sync does not write loads from those sources into a beam. Do not expect a transfer.

6.9.2 Out of a beam (the OUT dots)

Each design has an OUT dot "Support reaction G_k / Q_k". It goes to a **Column** (and to **Math**). The reactions are characteristic (unfactored) and split by class, solved from the same direct-stiffness model: G and Q separately, one row per support. On the wire you choose:

- **Which support**: the governing (largest total) reaction by default, or any named support S1, S2 and so on;
- **What it becomes on the column**: axial P/Z (default, column below the beam), or a shear V_x or V_y;
- **Connection eccentricity** (mm) and the axis of the moment it produces;
- **Send the end moment** (for a fixed support, G_k and Q_k end moments M_G and M_Q).

The column then factors the G_k and Q_k values itself (Chapter 4). A wire that names a design that has since been deleted reports the problem instead of sending a stale number. An unsolvable beam sends nothing and says why.

A beam does not feed a Foundation or Base plate directly; go through a Column. Beams are not wired to each other.

6.10 Results, reports and the node card

- **Node card**: lists each design with its member, governing check and utilisation (utilisation above 85 % reads amber, above 100 % red).
- **Working panels**: collapsible cards give the substituted numbers for the combinations, analysis (factored loads, reactions, equilibrium and envelope), flexure, shear, crack control, cover, fire, anchorage, deflection (concrete) or classification, shear, bending, LTB for every restraint segment, web bearing for every patch, and deflection for every span (steel). A closed-form check of the diagram values is included.
- **Report**: the calculation note is built by the shared module-report builder, so it has the same layout as every other module (Chapter 14 describes the window and formats: Word, PDF, copy, text; four colour schemes; it re-reads the

live design each time). Contents:

- cover with PASS / FAIL and the governing check;
- summary of checks;
- Figures: applied loads and supports, shear force, bending moment, deflected shape and the proposed RC cross-section or steel section with restraint layout;
- model, supports, loads, reactions, peak envelope;
- actions and combinations, analysis working;
- the EC2 or EC3 design with its working basis; anchorage and lap lengths; cover, curtailment and end anchorage; LTB, web bearing and deflection per segment for steel;
- quantity and carbon with charts, and references (BS EN 1992-1-1, BS EN 1993-1-1 and UK NAs, BS 8500-1, EN 1992-1-2, EN 1993-1-5, SCI P364, Blue Book).

- The report quotes only what the design produced. With a design switched off it says "analysis only".

6.11 Tips, limitations and known caveats

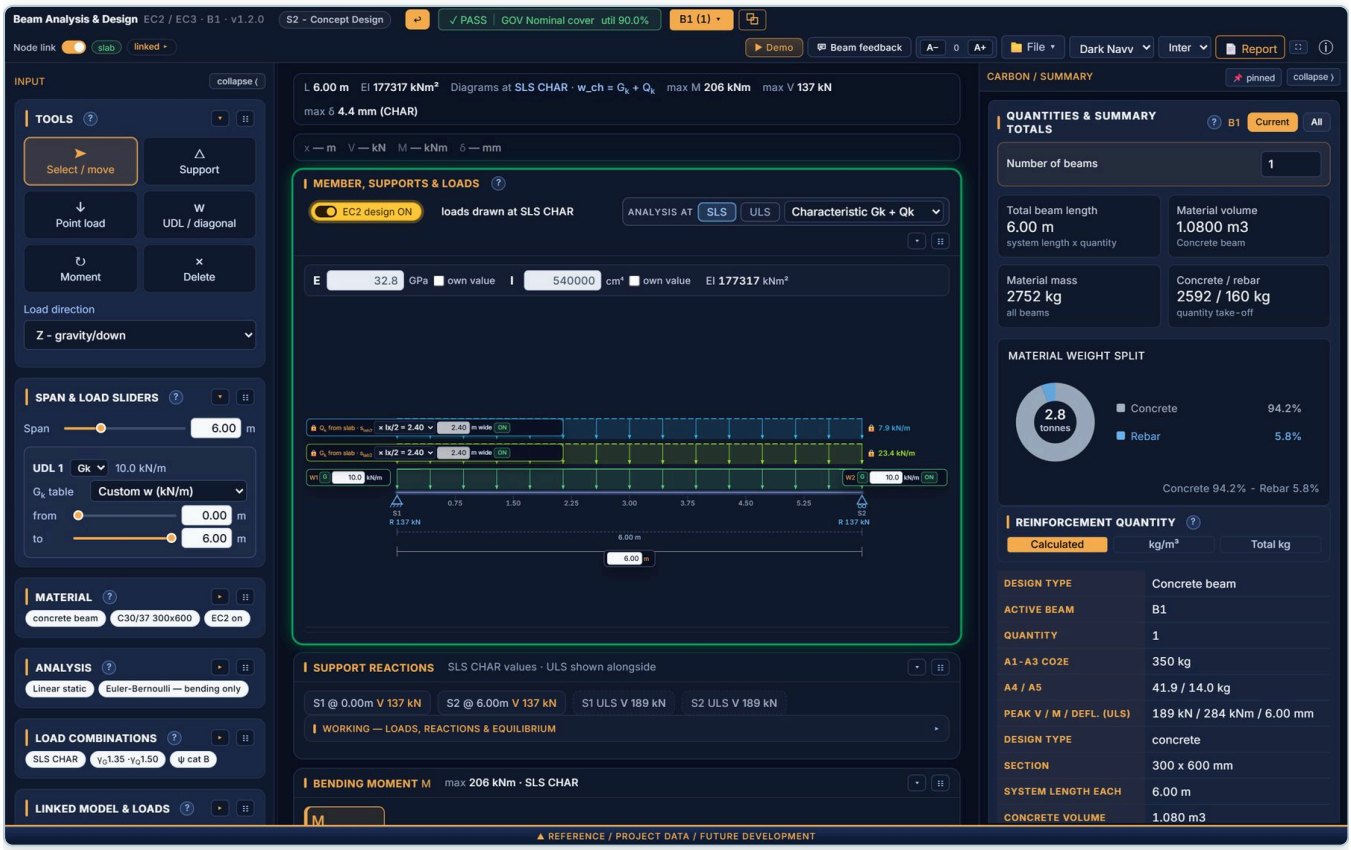
- **Defaults are a starting point.** A new beam is a 6 m simply supported concrete beam, 300 × 500 C30/37, 10 kN/m permanent plus its 3.75 kN/m self-weight, read at SLS. It passes with a governing "Nominal cover" at about 90 %, which is the auto-cover rule rather than real stress.
- **Read at SLS, design at ULS.** The drawn bending moment is service by default and so is smaller than the design moment. Switch to ULS to read the design actions.
- **Peak shear is used unreduced** in the EC2 shear check. Loads near a support are not credited.
- **Timoshenko off by default.** Turn it on for short deep members and transfer beams, and note the limitation on a fully built-in member.
- **Non-linear and vibration are not implemented**, despite appearing on the Analysis card.
- **Timber is not designed.**
- **Steel: class 4, shear buckling, biaxial bending, torsion and composite action are out of scope** and are flagged.
- **Web bearing is not counted by default** (connection mode). Change it if the beam really bears on a flange.
- **Torsional restraint at supports** is assumed (fork ends).
- **Concrete moments are elastic.** There is no moment redistribution of the analysis; δ only changes K' and the service stress.
- **Section sizes are not optimised automatically**; use the two optimiser buttons.
- **Deflection K for a fixed single support** reads cantilever (0.4).
- The creep coefficient in the explicit concrete deflection is fixed at 2.0 here (the Slab node, Chapter 8, derives it from humidity and age).
- A change that does not change the numbers does not redesign. Results are kept, so cards and maps stay light (searches for alternatives run only on a button).

> **References.** The standards, guidance and data sources used in this chapter are listed, with web links, in Chapter 17.

Screen reference - panel by panel

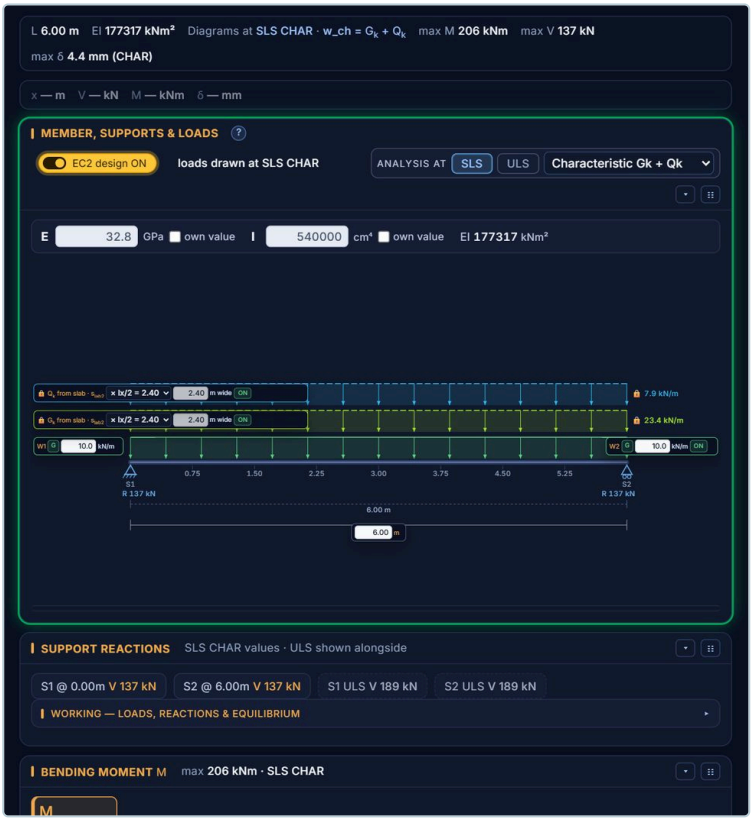
These screenshots show every panel of the module exactly as it appears on screen, scrolled from top to bottom, so you can match any field, button or result to the explanations in this chapter. Cards that are collapsed by default are shown expanded.

Reinforced concrete beam: the whole screen

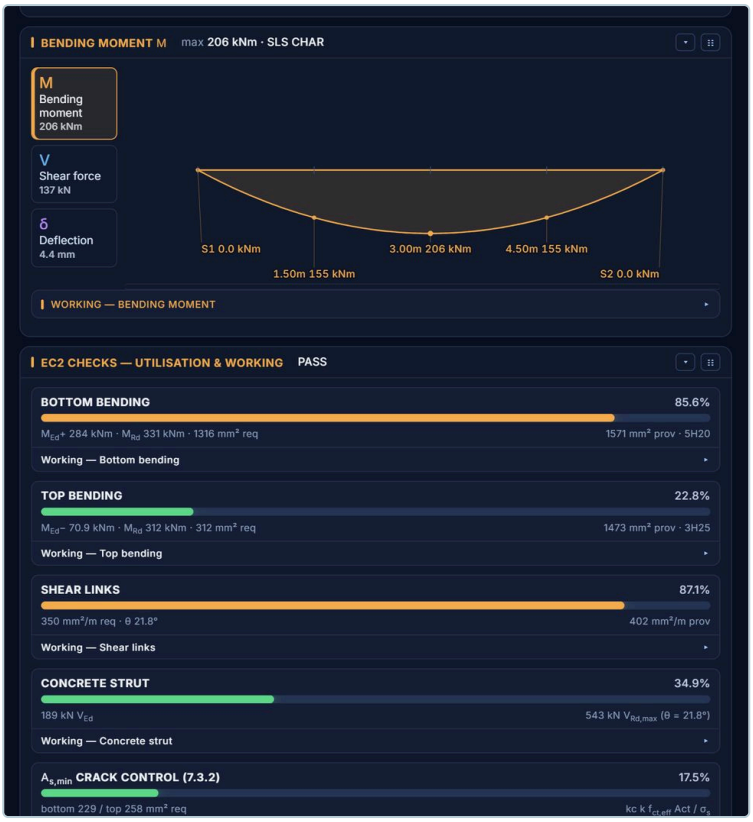


Reinforced concrete beam: the whole screen.

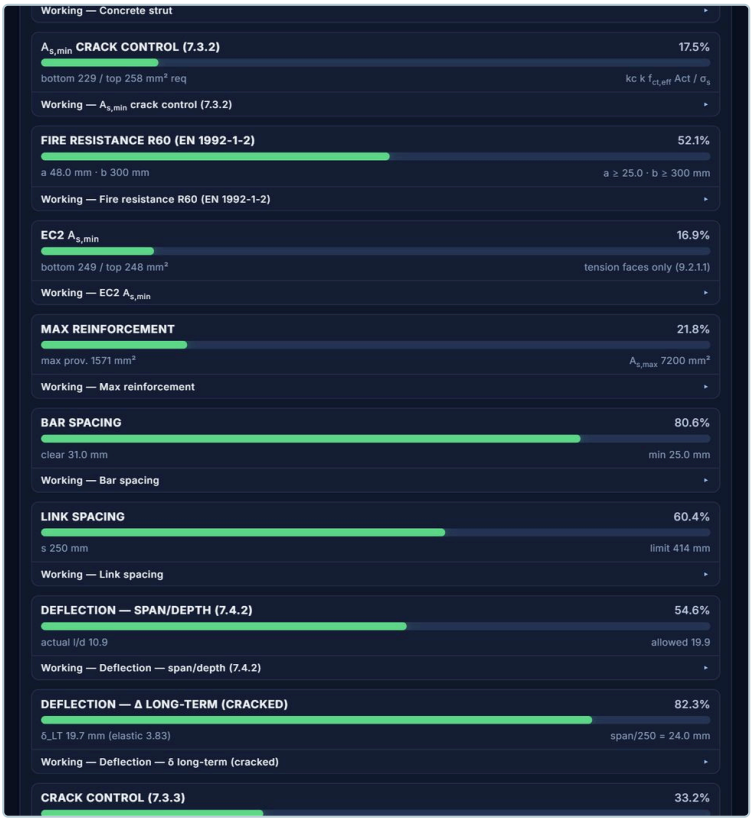
Reinforced concrete beam: working panel (centre of the screen)



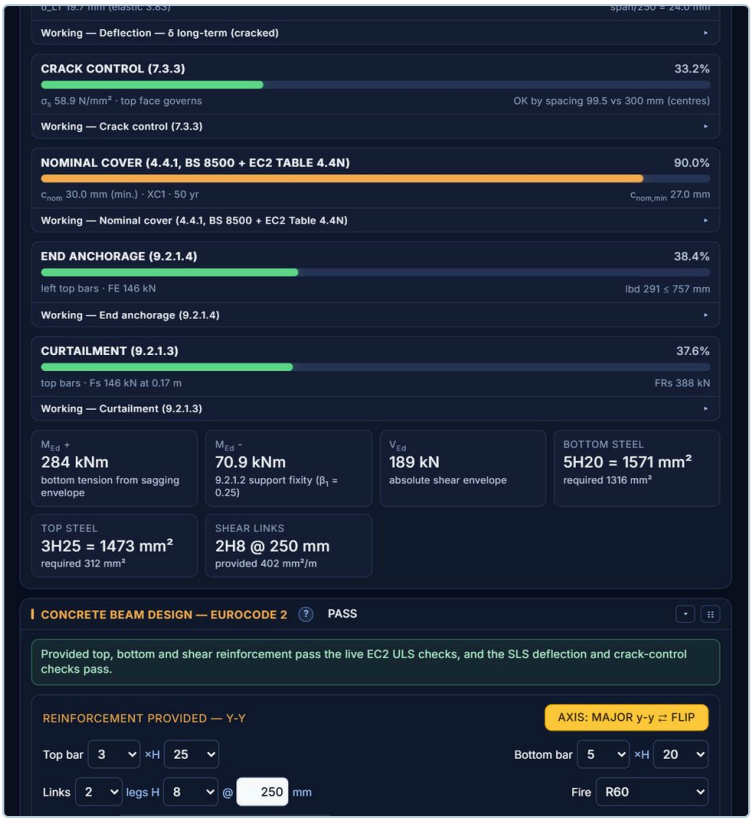
Reinforced concrete beam, working panel (centre of the screen), part 1 of 6. Cards shown: SLS char, Member, supports & loads.



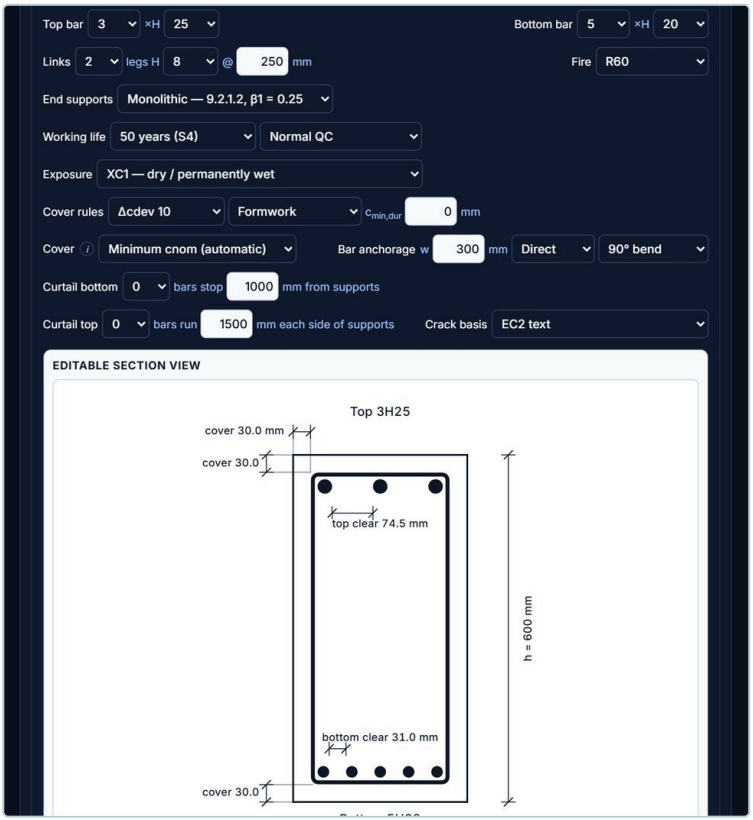
Reinforced concrete beam, working panel (centre of the screen), part 2 of 6. Cards shown: Bending moment m, Working — bending moment, EC2 checks — utilisation & working pass, Bottom bending, Top bending.



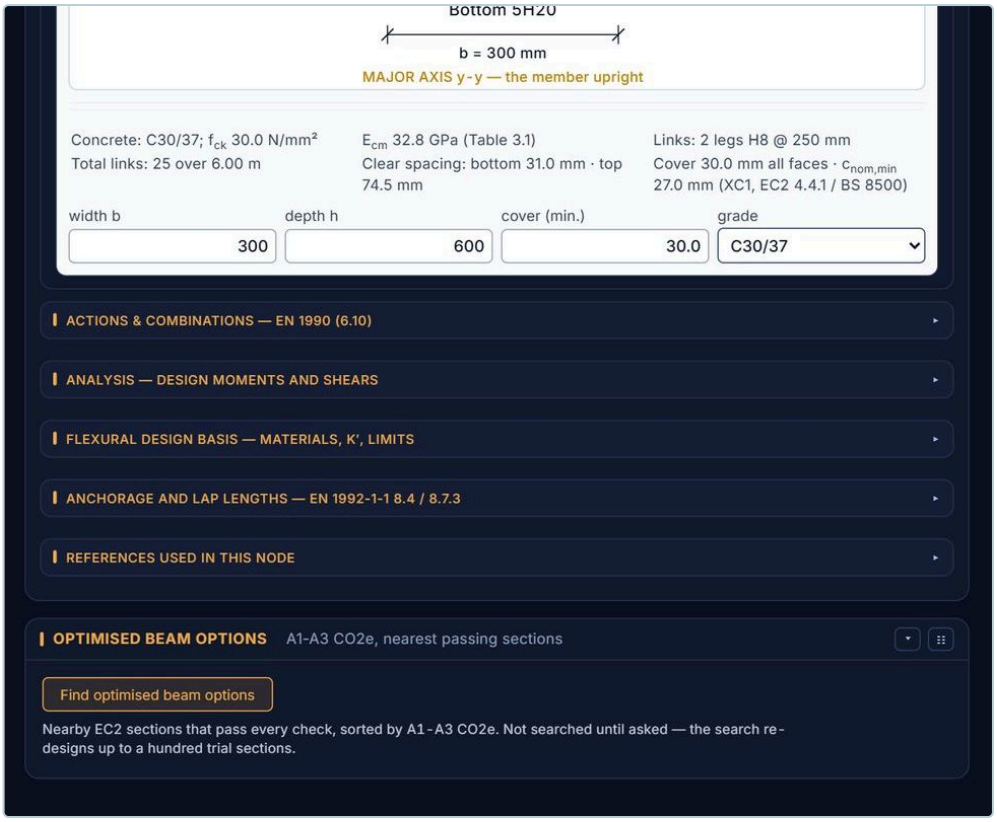
Reinforced concrete beam, working panel (centre of the screen), part 3 of 6.



Reinforced concrete beam, working panel (centre of the screen), part 4 of 6. Cards shown: Concrete beam design — Eurocode 2 pass.

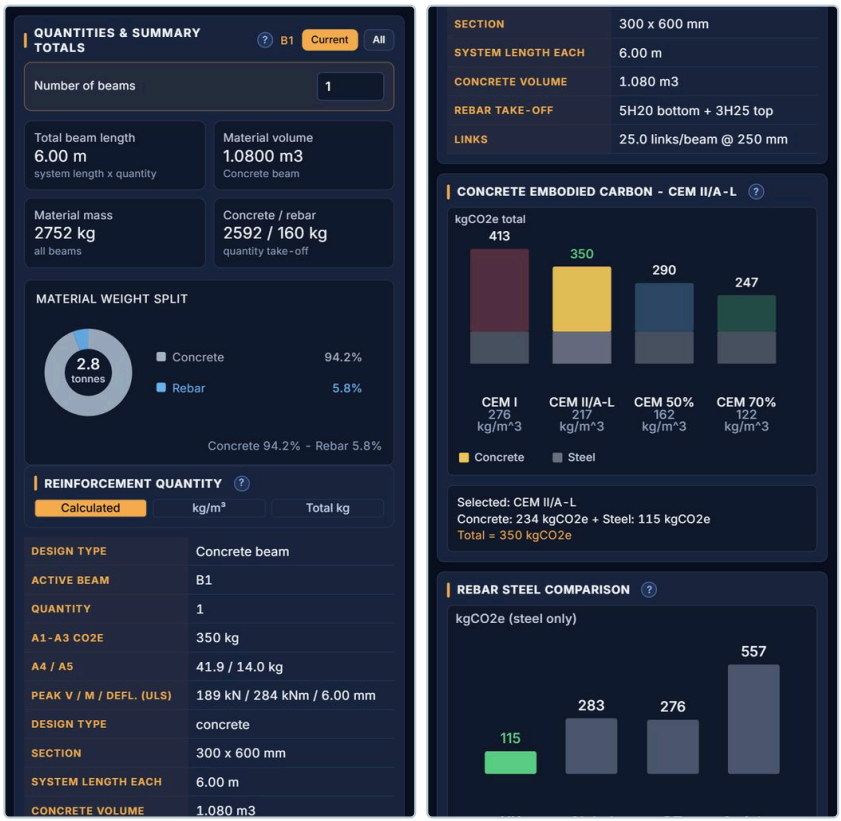


Reinforced concrete beam, working panel (centre of the screen), part 5 of 6. Cards shown: Editable section view.

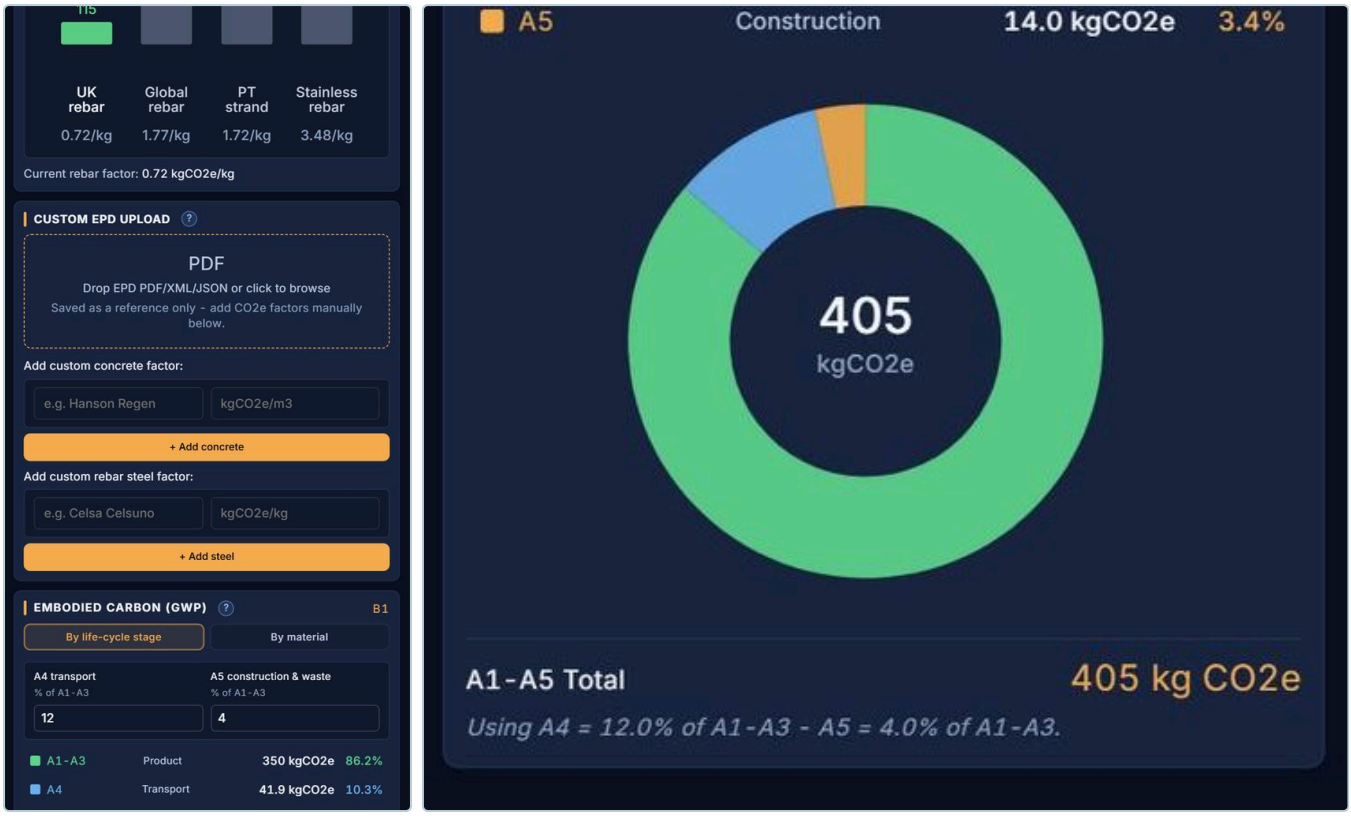


Reinforced concrete beam, working panel (centre of the screen), part 6 of 6. Cards shown: Analysis — design moments and shears, Flexural design basis — materials, k', limits, References used in this node, Optimised beam options.

Reinforced concrete beam: results panel (right of the screen)

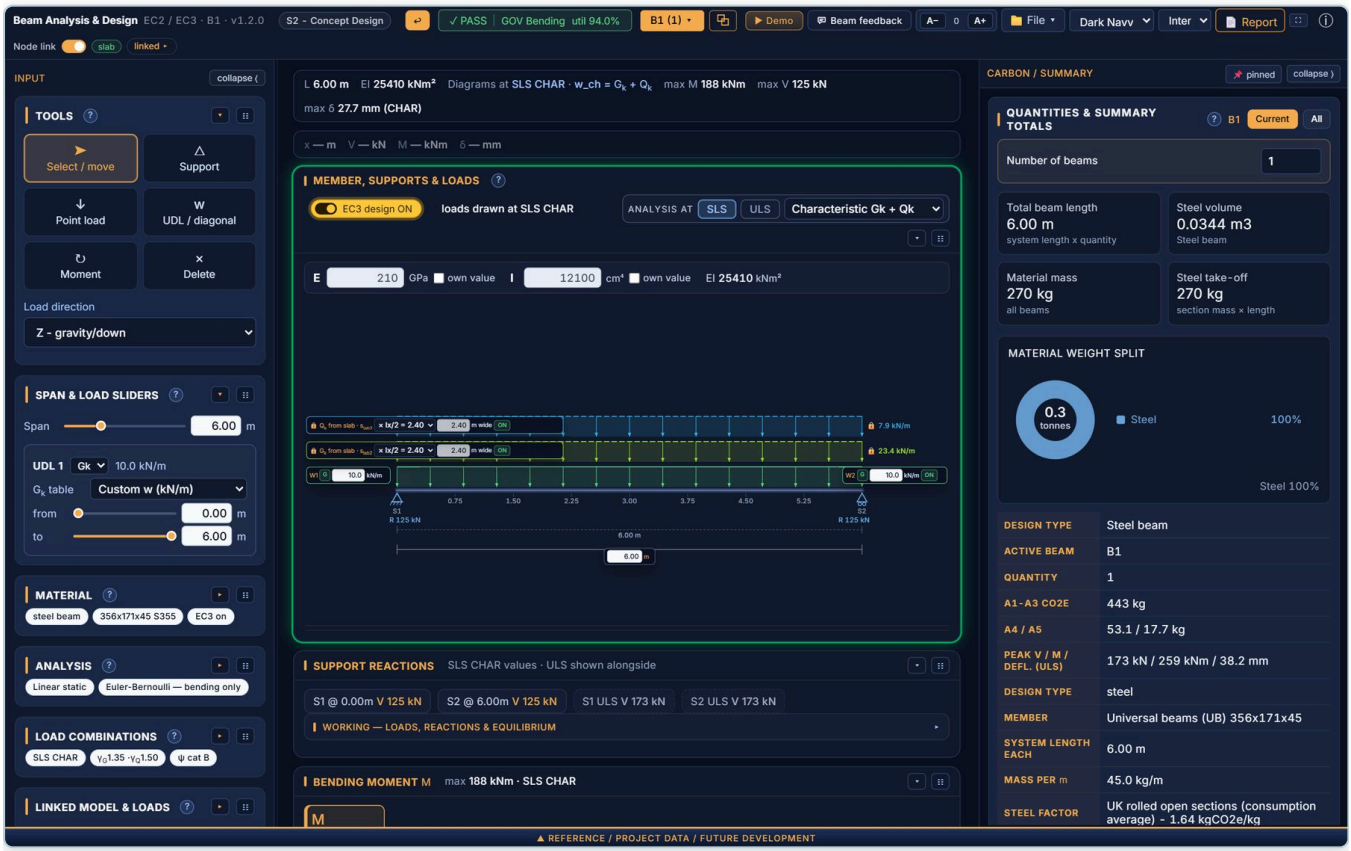


Reinforced concrete beam, results panel (right of the screen), parts 1-2 of 4. Cards shown: Reinforcement quantity, Concrete embodied carbon - CEM II/A-L, Rebar steel comparison.



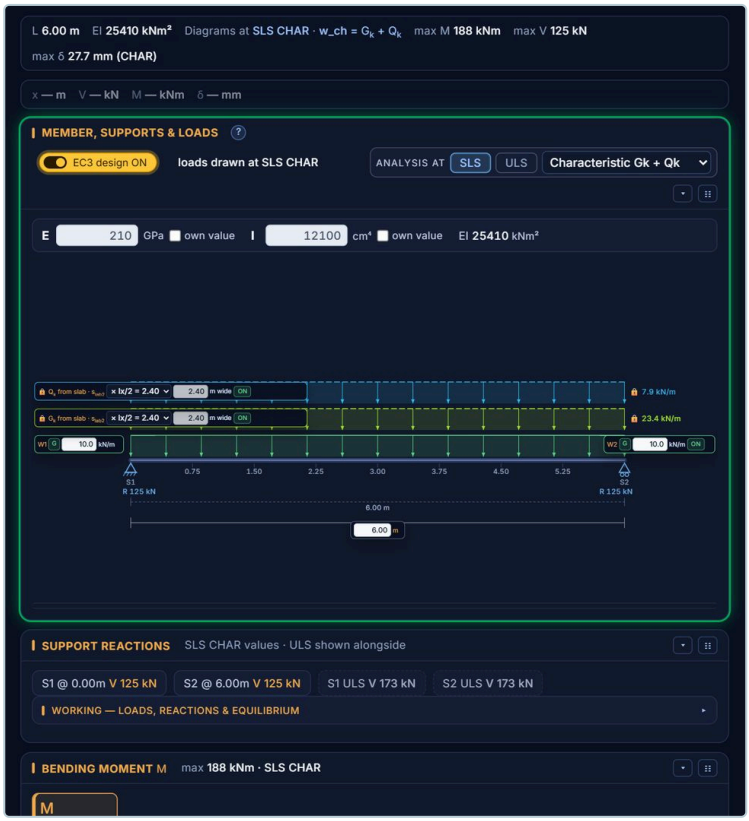
Reinforced concrete beam, results panel (right of the screen), parts 3-4 of 4. Cards shown: Custom EPD upload, Embodied carbon (GWP) B1.

Steel beam: the whole screen



Steel beam: the whole screen.

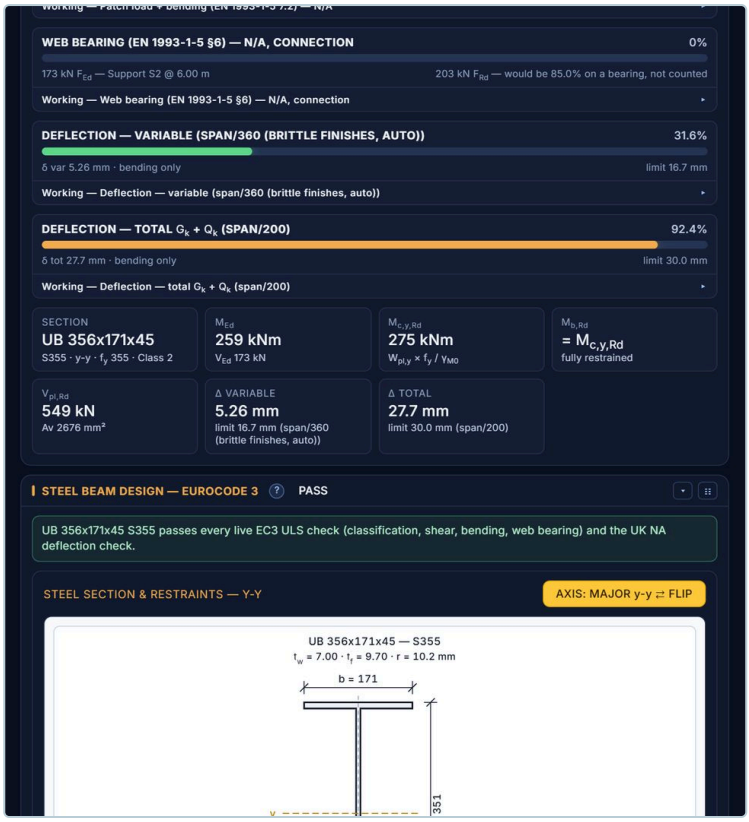
Steel beam: working panel (centre of the screen)



Steel beam, working panel (centre of the screen), part 1 of 5. Cards shown: SLS char, Member, supports & loads.



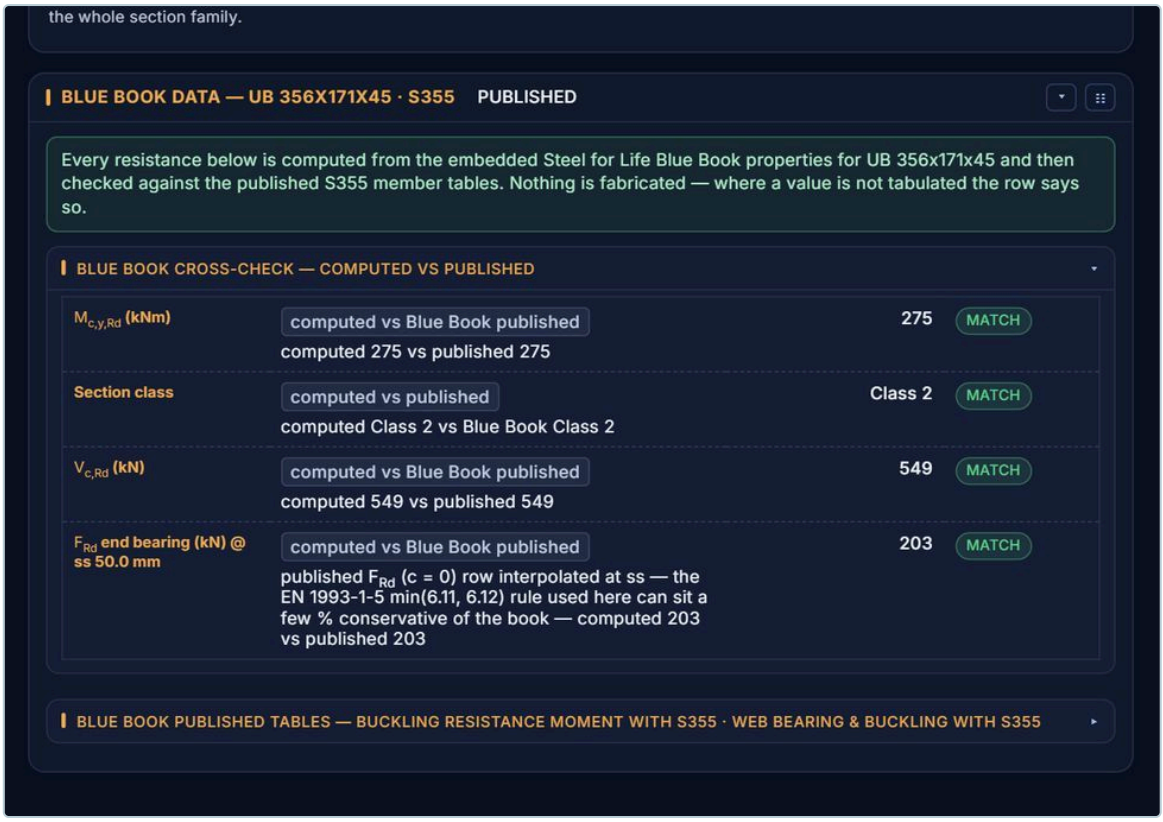
Steel beam, working panel (centre of the screen), part 2 of 5. Cards shown: Bending moment m, Working — bending moment, EC3 checks — utilisation & working pass.



Steel beam, working panel (centre of the screen), part 3 of 5. Cards shown: Steel beam design — Eurocode 3 pass, Deflection — variable (span/360 (brittle finishes, auto)).

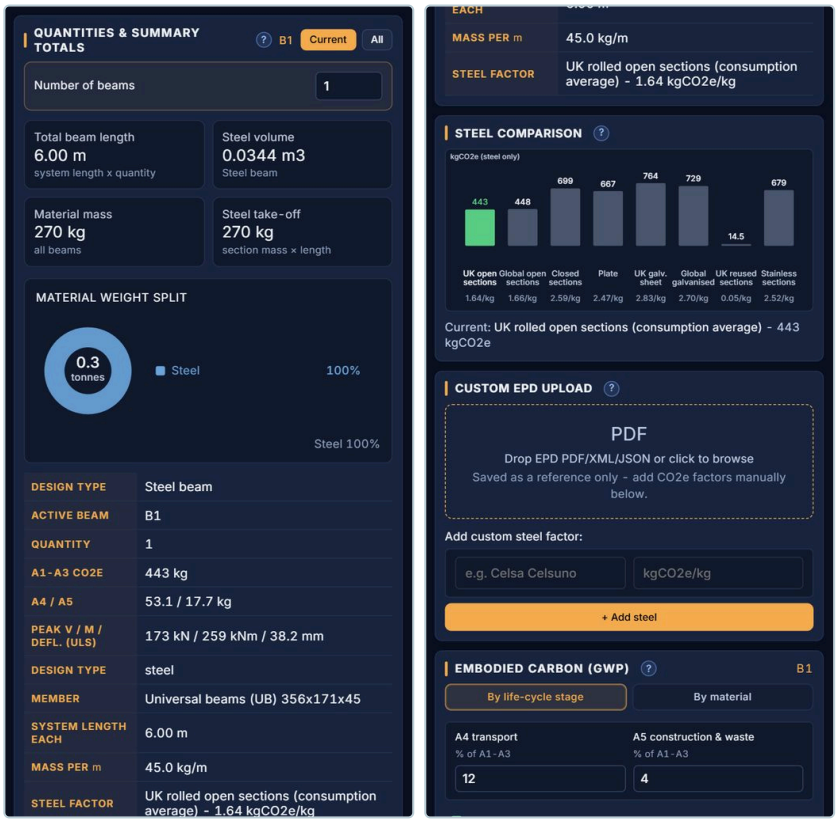


Steel beam, working panel (centre of the screen), part 4 of 5. Cards shown: Analysis — design moments and shears, Section & material basis — Blue Book + EN 10025, References used in this node, Optimised steel options.



Steel beam, working panel (centre of the screen), part 5 of 5. Cards shown: Analysis — design moments and shears, Section & material basis — Blue Book + EN 10025, References used in this node, Optimised steel options, Blue Book cross-check — computed vs published.

Steel beam: results panel (right of the screen)



Steel beam, results panel (right of the screen), parts 1-2 of 3. Cards shown: Steel comparison, Custom EPD upload, Embodied carbon (GWP) B1.



Steel beam, results panel (right of the screen), part 3 of 3. Cards shown: Embodied carbon (GWP) B1.

CHAPTER 7

Steel sections, Loads, Dimensions and Math nodes

Four small nodes do not design anything. They hold data, or do arithmetic on it, and pass the answer on by wire to the design nodes:

Node	Tag	What it holds	Feeds
Steel sections	Blue Book	A section from the Steel for Life Blue Book, with its properties and published resistance tables	Column (the map also permits Base plate, Foundation and Math, see 7.1.6)
Loads	Input	Named load sets: rows of dead, imposed and horizontal loads with unit conversion and a total	Beam, Slab, Foundation, Math
Dimensions	Geometry	A length, breadth and depth, with area and volume	Math
Math	Calc	One operation on two operands, carrying units	Beam, Slab, Foundation, Math

This chapter describes what each node does as built in v1.2.0. None of them has a pass/fail status, a report or a carbon figure of its own. Chapters 4 and 5 (Column, Base plate), 6 (Beam), 8 (Slab) and the Foundation chapter explain how the receiving nodes use what these nodes send. Chapter 2 covers how wires, ports and sync work.

7.1 Steel sections (Blue Book)

7.1.1 What it does

The Steel sections node is a window onto the Blue Book data embedded in the app. You pick a section family, a section and a **data route**, and the card shows that route's table for the section. A wire sends the section to a Column, Base plate or Foundation so that those nodes use the same section, and to a Math node.

The data are imported from Steel for Life Blue Book Excel exports. Nothing is looked up online. The node card has an "Open Blue Book" link to steelforlifebluebook.co.uk, which opens in a new tab.

7.1.2 Adding it

- Add **Steel sections** from the node library (tag "Blue Book").
- It is also created for you: when a Column is in steel mode and no Steel sections node exists, the app adds one and wires it to the column.
- Selecting the card marks it as the active steel data node, which also drives the lower data panel of the node map.

7.1.3 Inputs

Input	Options	Default
Family	See the table below	Universal beams (UB)
Section	Any section in the family, written as sizes and mass per metre, e.g. 356x171x45	First UB in the list (1016x305x584)
Data route	The routes available for the family (7.1.5)	Section properties (Detailing & fire is its own route)

The default route text is stored as "Section properties - Detailing & fire parameters"; the app maps the old text to "Section properties - Dimensions & properties" and offers "Detailing & fire parameters" as a separate route.

Changing the family resets the section to the first one in that family and the route to the family's first route.

7.1.4 Section families

14 families are offered, with the number of sections in each:

Family	Sections
Universal beams (UB)	107
Universal columns (UC)	46
Universal bearing piles (UBP)	17
Parallel flange channels (PFC)	16
Equal leg angles (L)	42
Unequal leg angles (L)	39
Back-to-back equal angles (L)	33
Back-to-back unequal angles (L)	32
Tees split from UC (T)	22
Tees split from UB (T)	68
Hot-finished circular hollow sections (CHS)	103
Hot-finished square hollow sections (SHS)	123
Hot-finished rectangular hollow sections (RHS)	161
Hot-finished elliptical hollow sections (EHS)	11

Where this node feeds a Column, only the family and section are used by the column (Chapter 4). The Beam node has its own section picker (UB and UC only) and is **not** a wire target of this node.

7.1.5 Section properties and resistance routes

Dimensions and properties route. For an I-section, the properties held per section include: mass per metre M, depth D, breadth B, flange thickness T, web thickness t_w, root radius r, depth between fillets d, ratios c_w/t_w and c_f/t_f, area A, second moments I_y and I_z, radii of gyration i_y and i_z, elastic moduli $W_{el,y}$ and $W_{el,z}$, plastic moduli $W_{pl,y}$ and $W_{pl,z}$, torsion constant I_t, warping constant I_w, and the buckling parameter U and torsional index X. Angles, channels, tees and hollow sections carry the equivalent set that applies to them.

Detailing & fire parameters route (UB, UC, UBP, PFC, tees). Detailing dimensions C, N and n, the surface area per metre and per tonne, and the section factors named in the data as three- and four-sided profile-protected and box-protected values. For 356x171x45 the data give $C = 6$, $N = 94$, $n = 20$, surface 1.36 m²/m and 30.2 m²/t.

Published resistance routes. The node reproduces the Blue Book tables for the chosen grade. Which routes appear depends on the family:

Family	Routes offered
UB	Dimensions and properties; Detailing & fire; Effective section properties (compression) S275, S355, S460; Axial compression S275, S355, S460; Buckling resistance moment S275, S355, S460; Web bearing & buckling S275, S355, S460; Axial force & bending S275, S355, S460
UC	As UB without the effective-section route
UBP	As UB; effective section routes for S355 and S460 only
PFC	As UC, plus "Axial compression connected web" for each grade
Equal angles	Dimensions; Detailing & fire; Effective section (compression) S275, S355, S460; Axial compression and Axial tension, S275, S355, S460

Family	Routes offered
Unequal angles	The above for S355, with compression and tension tables for short-leg-attached and long-leg-attached
Back-to-back angles	Dimensions and properties only
Tees (UC, UB)	Dimensions and properties; Detailing & fire
CHS	Dimensions; Axial compression, Bending resistance, Axial force & bending, for S355 and S420
SHS	Dimensions; Effective section (compression) and (bending y-y); Axial compression; Bending resistance; Web bearing & buckling; Axial force & bending, S355 and S420
RHS	As SHS, with the effective section in bending about both y-y and z-z
EHS	Dimensions; Axial compression; Bending resistance; Axial force & bending, S355

The grade is read from the route name.

The tables show a resistance against a length, for example $N_{b,y,Rd}$, $N_{b,z,Rd}$ and $N_{b,T,Rd}$ against buckling length for the compression route, and $M_{c,y,Rd}$, $M_{c,z,Rd}$ and $M_{b,Rd}$ by length and C_1 for the buckling resistance moment route. If a table is not in the data for a section and grade, the card says "No ... table found". Nothing is invented.

7.1.6 Outputs and ports

The node has no input. The wire menu lists these transfer fields: section data, dimensions, section properties, detailing/fire, compression tables, LTB / $M_{b,Rd}$, web bearing / buckling, axial force + bending, effective compression, and mass / CO_2. The wire sends **all** the selected Blue Book data (the wire menu says "no manual field checkboxes are required"). The wire carries the section, family, route, grade, the property set and the tabulated design data. What the receiver does with it differs:

Receiver	What happens
Column (steel)	Reads the family and section (Chapter 4, 4.3.1). A connected Steel sections node overrides the column's own family and section, and the column says so. If Auto sync is off, the column reads the section held in the wire's last synced snapshot, so edits reach it only on Auto sync or Sync now. Until a wire has synced, the column keeps its own section.
Base plate, **Foundation**	The map permits the wire, but the base plate and foundation do not use this section. Treat it as column-only unless you see the section arrive in the target.
Math	Permitted, and the section name is shown, but the arithmetic reads it as having no numeric value (0). No property is extracted as a number.

With several columns, wire a Steel sections node to the specific column input that should use it. A section chosen on an unlinked column stays that column's local trial and does not alter another column's Steel sections node. If an old "main" wire exists and no specific wire has been drawn, the column follows the first Steel sections node.

7.1.7 Tips and limitations

- The node is a **data source only**. It checks nothing. Use a Column, Base plate or Beam node for resistances.
- The Beam node cannot be wired from it; select the beam's section on the beam itself.
- Tabulated resistances are in the Blue Book tables for specific grades and lengths. The column node interpolates them; this node only displays them.
- For hollow sections the grade is S355 or S420; other families S275, S355 or S460.
- Data are as imported. The app does not update them if the published Blue Book changes.

7.2 Loads node

7.2.1 What it does

The Loads node is a spreadsheet-style take-off. It holds one or more **load sets** (tabs). Each set is a list of rows. A row picks a load from a library or a typed value, converts it through an area, a width or length-breadth-height dimensions, and gives a resultant in kN, kN/m or kN/m². The resultants are summed and sent by wire to a Beam, Slab, Foundation or Math node. The **active tab** is the one that is sent. Switching tab changes what the wires carry.

7.2.2 Adding and using it

- Add **Loads** from the node library (tag "Input"). The card is wide and shows its tabs, rows, total and reference list.
- New nodes start with one set named "Set 1" and no rows.
- Tabs: click a tab to make it active; double-click it (or use the edit box) to rename; **+** adds a set; the **x** on a tab deletes it after a second click ("sure?"). The last set cannot be deleted.
- Add rows with **+ Add load**; delete a row with its x.
- Typing in a row updates wired slab and beam cards as you type. Menu-type changes (preset, class, area/width ticks, add/delete) sync when the edit finishes. The usual **Auto sync** / **Sync now** gating applies (Chapter 2).

7.2.3 Row fields

Field	Meaning	Default
Name	Free text. If left blank the preset's name is shown	blank
Class	**G** permanent, **Q** variable (imposed), **H** horizontal	G
Direction	Fz, Fx or Fy of the resultant in the output vector	Fz
Σ tick	Include this row in the set total	on
Preset	The load library (7.2.4) or a custom entry	Custom value (kN/m²)
Value	The number for the chosen preset kind	1
Dims	L, B, H (density rows only), in m or mm	blank
area -> kN	For area-load rows: multiply by $L \times B$ (m or mm) or by a directly entered area	off
width -> kN/m	For area-load rows: multiply by a loaded width	off
Answer unit	Display the answer in the kN family (kN, kN/m, kN/m²) or the N family (N, N/mm, N/mm²)	kN

Class gives the sub-total used downstream: G is **DL**, Q is **LL**, H is **HL**. Changing the class resets the preset (Q to category B1, others to a custom value).

7.2.4 Libraries and the unit chain

Presets by class:

- **G (permanent)**: custom value (kN/m²); direct load (kN); custom density ($\mathrm{kN/m^3} \times L \times B \times H$); the dead-load library of CCIP-041 Table 2.14 (precast units, ribbed and waffle slabs, finishes, plasterboard, ceilings, roofing, tiles and similar, in kN/m²); and the bulk densities of CCIP-041 Table 2.13 (concrete, screed, brickwork, timber, steel, stone and similar, in kN/m³).
- **Q (variable)**: custom value, direct load, custom density, and the EN 1991-1-1 / UK NA imposed-load categories (A1 to H) with their q_k in kN/m² from CCIP-041 Tables 2.2 to 2.8. Storage rows E13, E15 to E19 are tabulated per metre of storage height and are priced here at a 2.5 m stack; type a custom value for another height.
- **H (horizontal)**: custom value, direct load, custom density.

Unit chain. Each preset has a kind, and the answer's unit follows from it:

Row kind	Start unit	Conversion	Result
Area load (kN/m²)	kN/m²	none	kN/m²
		tick "area -> kN" ($L \times B$, or a direct area)	kN
		tick "width -> kN/m" (width)	kN/m
Direct load	kN	none	kN
Density (kN/m³)	kN/m³	multiply by each of L, B, H that is filled	one dimension: kN/m²; two: kN/m; three: kN; none: kN/m³

So "Concrete - reinforced, 25 kN/m³" with L, B and H filled in is a self-weight in kN; with L and B filled it is a load per metre; with only a depth it is an area load. The two ticks are exclusive. A missing dimension shows a red error in the row ("enter L x B or a direct area", "enter the loaded width").

Σ **total.** The sum of the ticked rows. It is only valid when all ticked rows share the same unit. Otherwise the total shows "UNITS ERROR - mixed ... Untick Sigma on mismatched rows."

7.2.5 What a wire carries

A load node has these output dots:

Dot	Carries
DL	Sum of the G rows of the active set: forces Fx, Fy, Fz in kN, plus G line (kN/m) and G area (kN/m²) sums
LL	The same for Q rows
Horizontal (HL)	Forces of H rows. At the receiver these are the wind-type (WL) fields
Σ	One answer: the total of the ticked rows, in whatever unit they share
One dot per row	That row's answer exactly as it reads

(The H class puts its line and area sums into the Q totals, since only forces have a separate HL channel.) Moments are always zero from the set; the take-off model produces forces and loads only.

A row dot or Σ dot bypasses the DL / LL / HL field choice on the wire menu: it sends exactly that answer.

7.2.6 What the receivers do

Receiver	Behaviour
Beam	Line rows (kN/m) become full-span UDLs, split G and Q. Vertical force rows (kN, Fz) become mid-span point loads (DL as G; LL and HL as Q). Area rows become UDLs through a loaded width that starts at 1 m and can be changed on the beam (Chapter 6).
Slab	Only **area** loads in kN/m² are accepted. Rows in kN or kN/m are rejected with the reason shown. The wire chooses which slab load field a class drives: screed, finishes and services, movable partitions or imposed qk (defaults: permanent to finishes, variable to qk). Ticking several fields makes one value cover them all, written to the first and zeroing the others to avoid double counting. A sending wire locks the slab fields it fills; the slab's Loads panel has a switch to turn all wires off and edit by hand (Chapter 8).
Foundation	DL, LL and HL vectors go to the DL, LL and WL (horizontal) load rows, as locked loads.
Math	Used as an operand with its unit (7.4.3).

A Loads node cannot be wired to a Column or Base plate; take the loads through a Beam or Slab, or type them into the column.

7.2.7 Self weight and older load nodes

There is no separate self-weight input in the current model. Enter self weight as a G row: for example "Concrete - reinforced" (25 kN/m³) with the L, B and H of the element, which gives kN. Older saved load nodes (single DL / LL / HL Fx, Fy, Fz, Mx, My fields and a named self weight) are converted on opening into rows of Set 1; the self weight becomes a G row named as it was. The converted node loses moment entries (Mx, My), because the row model carries forces only.

7.2.8 Reference documents

A card can hold references: PDFs or images. Images up to 400 kB are stored inside the project as data. Anything else (PDFs and larger images) is stored by name only; the note "reference only - file not stored" is shown, and the file must be kept separately.

7.2.9 Tips and caveats

- Check class: an imposed load typed as G will take 1.35 rather than 1.5 downstream.
- Use the answer-unit selector to read in N if your source data are in N; the inputs stay as typed.
- A beam or slab fed by a load set shows locked bands. Use "Type my own loads instead" on the beam to detach.
- The Σ and row dots send the unit of the answer. The slab refuses anything but kN/m², the beam accepts all three units, and a foundation takes forces.

7.3 Dimensions node

7.3.1 What it does

A tiny geometry holder. It carries three lengths and derives an area and a volume, so that a Math node can combine them with loads (for example a pressure times an area).

7.3.2 Inputs and outputs

Field	Unit	Default
Name	text	Dimensions
L	m	1.0
B	m	1.0
Depth D	m	1.0

The card result reads "Area / volume": $\text{area} = L \times B$ (m²) and $\text{volume} = L \times B \times D$ (m³). Four output dots, each carrying one number with its unit, each printing its live value:

Dot	Value
L	Length, m
B	Breadth, m
Area	$L \times B$, m²
Volume	$L \times B \times D$, m³

There is no combined $L \times B$ dot and no main output socket, because a mixed vector would have no single unit.

7.3.3 Connections

A Dimensions node can be wired **only to a Math node** (as operand A or B). It takes no input. It cannot be wired directly to a foundation, beam, slab or column: use Math, with the unit algebra of 7.4.3.

7.4 Math node

7.4.1 What it does

Math performs one operation on two operands, A and B, and gives a result with a unit. Operands are typed numbers or wired values: a load row, a DL or LL dot, a Dimensions output, another Math result, a Blue Book property, or a design output such as a beam reaction or a column load.

7.4.2 Inputs

Field	Options	Default
A	A number, or a wire	0
Operator	$+$, $-$, $\times$, $/$, $\max$, $\min$	$+$
B	A number, or a wire	0

A wired operand is locked: its box shows the live incoming value and ignores the typed one. Delink the wire to type a number again. Several wires into one operand are combined (summed) before the operation.

7.4.3 The calculation and units

Wired values are tagged with a unit string such as kN, kN/m², kg, m³ or kgCO₂e. The operator carries the unit through:

Operator	Unit rule
$+$ and $-$	Both sides must have the same unit. A plain number takes the other side's unit. A mismatch is flagged
$\times$ and $/$	Units combine algebraically: $\mathrm{kg/m^2}$ gives $\mathrm{kg/m^2}$; $\mathrm{kN/m^2} \times \mathrm{m^2}$ gives kN; $\mathrm{m^3/m^3}$ gives a plain number
$\max$, $\min$	Compare and return one operand

Tokens are matched as written. There is **no SI conversion**: kN against N is a mismatch. If the unit of the answer is unknown, targets that need a strict kN family unit (the slab) refuse it rather than assume kN/m².

Vectors. A load wire carries several components (DL, LL, WL forces). The rules:

- A plain number against a vector multiplies or divides only the **non-zero** components. A row answer scaled by a factor scales the answer.
- Adding, subtracting, $\max$ or $\min$ of a plain number against a multi-component load has no single meaning and is refused with a message.
- Two vectors combine component by component.
- For $\times$ and $/$, a Σ-total wire is used as one scalar factor ("row $\times$ Σ" means answer $\times$ total).

Errors. Division by zero is an error, not zero: the result carries "division by zero" and downstream nodes receive nothing meaningful from it. A cycle (a Math node feeding itself) returns zero and says "cycle". The card shows the answer text and a tooltip describing any error. An error on a Σ wire is shown in red when the wire is synced.

7.4.4 Output and wiring

One output dot, **Result**, with its live value printed beside it. It can be wired to a Beam, Slab, Foundation or another Math node. A Math node can take input from Loads, Dimensions, Math, Column, Foundation, Base plate, Beam, Slab, IFC, Wind and Sketch-ET (and, formally, Steel sections, but see 7.1.6: that gives 0).

Typical chains:

- **Pressure × area**: a Loads row (kN/m²) × a Dimensions Area (m²) gives kN, which a foundation accepts as an axial load.
- **Load / width**: a total in kN divided by a Dimensions L (m) gives kN/m for a beam.
- **Sum of two beams' reactions**: both reaction wires into A and B with +.

At the receiver, the transfer is placed as the unit says. A kN/m² answer into a beam is an area load; a bare number into a beam is a permanent UDL over the span; a kN answer into a slab is rejected.

7.5 Which wires are allowed

The map only accepts these pairs for the nodes in this chapter:

From	To
Steel sections	Column, Base plate, Foundation, Math (only the Column reads the wire, see 7.1.6)
Loads	Beam, Slab, Foundation, Math
Dimensions	Math
Math	Beam, Slab, Foundation, Math

Nothing can be wired **into** Loads, Dimensions or Steel sections. A Math node accepts wires from every design node. Where a pair is drawn but the sync does not apply it (Wind or Sketch-ET to a Beam, for example), the receiving chapter says so.

7.6 Tips, limitations and known caveats

- These nodes have **no result flags** and appear in no report. They show up in a design report only through the numbers they sent.
- The Loads node sends the **active** tab only. If a downstream number looks stale, check which tab is active.
- Unit errors are caught at the wire, not silently converted: a mixed-unit Σ, a kN/m wired into a slab or a division by zero all stop with a message.
- Keep class (G, Q, H) correct. Downstream nodes factor G and Q differently.
- Blue Book data are display and transfer only. Utilisations come from the nodes that use the section.
- Moments entered in an older load node are not preserved when it is converted to rows.
- The Steel sections node does not feed the Beam node.

> **References.** The standards, guidance and data sources used in this chapter are listed, with web links, in Chapter 17.

CHAPTER 8

Slab design

The Slab node designs a reinforced-concrete slab panel to EN 1992-1-1 (EC2) with the UK National Annex. In v1.2.0 it designs **one-way spanning** slabs and **two-way restrained** slabs, in-situ, with a live plan, section and design-diagram drawing, a 3D view with a reinforcement heat map, a full set of EC2 checks with the substituted numbers shown, and an embodied-carbon take-off. A slab's loads can come in by wire from a Loads node, a Math node or an IFC element, and its area loads can be sent on to a Beam or a Column.

Analysis is by a 2-D elastic plate finite-element solution by default, with the code coefficient tables (Concrete Centre "Table 3" for one-way, "Table 8" for two-way) kept alongside for comparison, and as the basis when the FE is switched off.

This chapter describes what the node does as built in v1.2.0. Clause numbers quoted are those used in the calculations and reports.

Not available in this release. The slab type list also contains **Flat slab** and **Ribbed slab**, and the design-system list contains a **Tata Steel ComFlor composite metal deck** option. All three are shown locked ("locked" / "under development") and cannot be selected; a saved design that carries one is opened as a one-way RC slab. This chapter does not describe them.

8.1 Opening and adding a slab

- Add **Slab design** from the node library (tag "EC2").
- **Double-click** the card (not on a button, box or drop-down) to open the full Slab workspace.
- One Slab node holds a list of designs (S1, S2, ...). A picker in the module header adds, renames, switches and deletes them. Each design has its own OUT dot on the node card and its own report.
- The workspace has:
  - **Left**: input cards (collapsible, re-orderable): Design system, Slab type, Geometry, Loads, Point and line loads, Analysis, Materials and cover, Reinforcement, Design options, and so on. The panel folds to a rail.
  - **Centre**: a live drawing (plan with the bar layout to scale, section A-A, and design diagrams for bending moment, shear force, deflection and required reinforcement), a **3D view**, the working calculations, and a **Reference** tab that explains the FE basis in detail.
  - **Right**: the quantities and embodied-carbon panel.
- Global Ctrl+Z / Ctrl+Y undo and redo cover the slab design.
- A header chip and the node card show PASS, WARN or FAIL, the governing check and its utilisation.

8.2 Inputs

8.2.1 Slab type, geometry and supports

Input	Meaning	Unit	Default
Slab type	One-way spanning, or two-way restrained	-	One-way
Span L	One-way: the span. Two-way: the short span l_x (l_x is always the shorter side; entering a longer L swaps them)	m	4.8 (0.5 to 30)
Panel width L_y	One-way: the width of the panel. Two-way: the long span l_y	m	6.0
Slab depth h	Overall depth	mm	175 (50 to 1000)
Support width t	Width of the supporting wall or beam	mm	200

Input	Meaning	Unit	Default
Span measured as	"Effective span leff entered directly", or "Support centres - compute leff"	-	l_{eff} entered directly
Support / continuity (one-way)	Simply supported; end span, pinned end support; end span, continuous end support; interior span	-	Simply supported
Panel case (two-way)	Nine cases, see below	-	Interior panel

When the span is measured between support centres, $l_{eff} = l_n + a_1 + a_2$ with $a = \min(h/2, t/2)$ (EC2 5.3.2.2); the l_{eff} used is shown beside the input.

Two-way panel cases (a continuous edge continues into the next panel; a discontinuous edge does not): interior panel; one short edge discontinuous; one long edge discontinuous; two adjacent edges discontinuous; two short edges discontinuous; two long edges discontinuous; three edges discontinuous (one long edge continuous); three edges discontinuous (one short edge continuous); four edges discontinuous (corners held down).

A two-way panel with l_y/l_x above 2 raises a warning that it should be treated as one-way (the coefficients are clamped at 2).

8.2.2 Loads

Load	Unit	Default	Class
Self-weight	kN/m²	$h \times 25\ \mathrm{kN/m^3}$ (live from the geometry, 4.375 for 175 mm)	permanent
Screed	kN/m²	1.0	permanent
Finishes and services	kN/m²	0.5	permanent
Imposed q_k	kN/m²	2.5 (category B1)	variable
Movable partitions	kN/m²	0.8 (preset: partitions up to 2.0 kN/m)	variable
Snow s_k (0 = none)	kN/m²	0	variable (second)
Site altitude (snow ψ)	normal / high	normal	-

- **Libraries.** Screed, finishes, partitions and imposed q_k each have a preset list. Imposed values are the EN 1991-1-1 / UK NA categories (CCIP-041 Tables 2.2 to 2.8); screed and finishes build-ups come from CCIP-041 Tables 2.14 and 2.15; movable partitions are treated as an equivalent uniform variable action (EN 1991-1-1 6.3.1.2(8)). Any preset can be replaced by a custom value.
- **Switching components off.** Every component (self-weight, screed, finishes, imposed, partitions, snow) can be muted from the study chips in the 3D view. Muted components are excluded from every calculation and from what the slab sends downstream.
- **Wires.** A wire from a Loads or Math node can drive screed, finishes, partitions or q_k (8.9.1). Driven fields are locked; the Loads panel has a switch that turns all incoming wires off so the fields can be edited by hand again.

8.2.3 Combinations (EN 1990)

Input	Default	Notes
γ_G	1.35	Choices: 1.35 (Exp. 6.10) or 1.25 (the worse of 6.10a and 6.10b)
γ_Q	1.5	
Imposed category	B (office)	ψ_0, ψ_1, ψ_2 follow EN 1990 Table NA.A1.1; editable with "custom"
ψ_0, ψ_1, ψ_2	0.7, 0.5, 0.3	Category A and B values

The ULS area load is $n = \gamma_G g_k + \gamma_Q q_k$ (+ snow, below). With $\gamma_G = 1.25$ both Exp. (6.10a) $= 1.35\, g_k + 1.5\, \psi_0 q_k$ and Exp. (6.10b) are evaluated and the larger governs. When 6.10a governs, the working copy uses $\gamma_G = 1.35$ and $\gamma_Q = 1.5\, \psi_0$ so that added point and line loads are factored on the same expression as the area load. The working states which expression was used. Older saved designs left on 1.25 are moved to 1.35 on opening.

Snow is a second variable action: the design takes the worse of "imposed leading plus $\psi_{0,s}\times$ snow accompanying" and "snow leading plus $\psi_0\times$ imposed accompanying", each at 1.5. Snow ψ is 0.5 / 0.2 / 0 at normal altitude and 0.7 / 0.5 / 0.2 for the high-altitude option. For a **Category H roof** the UK NA (EN 1991-1-1 3.3.2) rule applies: roof imposed load is not combined with snow, and the worse of each acting alone governs.

8.2.4 Point and line loads

For one-way and two-way panels with the plate FE on, you can add **point loads** and **line loads** (the + Point load / + Line load buttons), and drag them in the 3D view.

Field	Point load	Line load
Magnitude	P, kN (or pick a concentrated load Q_k from the EN 1991-1-1 category library)	w, kN/m
Position	x, y (m)	centre c_x, c_y (m); direction X or Y; length (m)
Loaded width	b, m (default 0.3)	b, m
Class	Q (variable) or G (permanent)	Q or G

Each load spreads at 45° through the slab depth, so the loaded patch is $b + h$. Q loads take γ_Q and ψ_2, G loads take γ_G. With the plate FE off, point and line loads are switched off with it, because a concentrated load cannot be dispersed by a coefficient table; stored loads are kept and return when the FE is switched back on.

8.2.5 Materials, cover and fire

Input	Options	Default
Concrete grade	C25/30, C28/35, C30/37, C32/40, C35/45, C40/50, C45/55, C50/60	C30/37
f_{yk}	N/mm²	500
Exposure class	BS 8500-1 Table A.4 classes	XC1, dry / permanently wet
Design working life	50 or 100 years	50
Special QC of the concrete	on/off	off
Cover mode	automatic minimum, or user-defined	automatic
Nominal cover, bottom / top	mm	25 / 25
$c_{min,dur}$ override	mm (0 = automatic)	0
Fire rating	REI 60, REI 90, REI 120, REI 240	REI 60
Bars lapped at one section	% (for α_6)	100
Bottom bars anchored at a simple support	%	50

The 50 % default for anchored bottom bars follows EC2 9.3.1.2(1), which guarantees half the span steel at a simple support. It sets the area A_{sl} used for ρ_l in the shear check there.

8.2.6 Reinforcement

Input	Default
Main bars, bottom (one-way; direction l_x of a two-way panel)	H12 at 175
Main bars, bottom, direction l_y (two-way)	H12 at 200
Top bars (over supports)	H12 at 175
Secondary (distribution) bars, one-way	H10 at 350

8.2.7 Design options and long-term inputs

Input	Default	Used for
Redistribution ratio δ	1.0	K' and the hogging service stress (see 8.3.4)
Ductility cap K' of 0.168	off	Optional cap on K'
Brittle partitions	off	Deflection factor F_2
Adjacent spans differ by up to	0 %	Warns when the continuous models' validity (spans within about 15 %) is exceeded
Relative humidity RH	50 %	Creep and shrinkage (inside conditions; use 80 % outside)
Age at first loading t_0	28 days	Creep
Cement class (EC2 3.1.2(6))	N	Creep and shrinkage
Rebar allowance (carbon)	110 kg/m³	Take-off (8.8)

8.2.8 Analysis settings (plate FE)

Input	Options	Default
Plate FE analysis	on/off master switch	on
Analysis basis	Full plate FE (2-D); code coefficients + plate FE for point and line loads; code coefficients + effective-width strip (1-D)	Full plate FE
FE mesh density	Coarse (12 divisions on the short span), Normal (18), Fine (26), Very fine (36)	Normal
Peak smoothing width	m (0 = automatic)	0
20 % support redistribution (UK NA 5.1.3(1)P option (c))	on/off	on
Punching check under point loads	on/off	on
Panel side edges (one-way)	slab carries on past them (symmetry), or free edges	symmetry
Spans in the run (one-way end span)	2 to 6	3

8.3 Analysis

8.3.1 Code coefficient tables

- **One-way (Table 3).** Moments and shears as multiples of $F = n \times l_{eff}$ per metre width, with 20 % redistribution at supports built in:

Condition	Span moment	End-support moment	Interior-support moment	Shear (end / interior)
Simply supported	$0.125\,Fl$	0	0	$0.50\,F$
End span, pinned end	$0.086\,Fl$	0	$0.086\,Fl$	$0.40\,F$ / $0.60\,F$
End span, continuous end	$0.075\,Fl$	$0.040\,Fl$	$0.086\,Fl$	$0.46\,F$ / $0.60\,F$
Interior span	$0.063\,Fl$	$0.063\,Fl$	$0.063\,Fl$	$0.50\,F$

(Simply supported is $nl^2/8$ and $nl/2$.)

- **Two-way (Table 8, BS 8110-1 Table 3.14 and shear Table 3.15).** $M = \beta n l_x^2$ for the short and long span, midspan and edge, with β interpolated on l_y/l_x over the full column set 1.0, 1.1, 1.2, 1.3, 1.4, 1.5, 1.75, 2.0 for the nine panel cases. Shear coefficients per edge type are used for the support shears.

These coefficients embed the UK NA option (c) arrangement (all spans loaded, 20 % redistribution). They are only valid inside the conditions of 8.3.3. Outside them the node adds a **failed** "Load arrangement - coefficient validity" check (not a warning), because arrangements (a) and (b) of 5.1.3 then govern and the coefficients do not cover them.

8.3.2 Plate finite element (default)

The panel is solved as a **Kirchhoff thin elastic plate** with 12-term ACM rectangular elements (3 degrees of freedom per node), plate stiffness $D = E_{cm}h^3/(12(1-\nu^2))$ with Poisson's ratio $\nu = 0.2$ and the uncracked section. The mesh follows the density setting and is capped at a number of nodes (400, 900, 1800 or 3200). The working prints the mesh, degrees of freedom and the last solve time.

- All loads, the area load and every point and line load, act in one solution, so each response depends on where the load really sits.
- Design moments are **Wood and Armer** moments, so twisting moment is included in the reinforcement demand in each direction.
- **Peak smoothing.** Thin-plate theory is singular under a point load, so the raw FE peak is averaged across a band before it becomes a design moment. The default band is the effective width $b_e = 0.3 + 2.4a(1 - a/l)$ (the width the code strip method uses), floored at $3h$. Where smoothing takes more than 15 % off the raw peak the working warns that a concentrated load is driving the panel and the band width matters. EC2 gives no rule for averaging elastic peaks; the Reference tab says so.
- **Supports.** One-way panels have two line supports (side edges symmetric or free). A one-way **end span** is modelled as a whole run (2 to 6 spans, default 3) on knife-edge internal supports and designed on bay 1, because clamping the first interior support would assume the next bay infinitely stiff and return $nl^2/8$ instead of the real continuous value of about $nl^2/10$ for three equal spans. Two-way panels take their continuous edges as clamped and their discontinuous edges as simple.
- **Shear** is the peak support reaction per metre run, averaged over a band of $3h$ (minimum 0.3 m).
- **Top steel in held-down corners.** Where a panel has corners held down, the top Wood and Armer twisting moment outside the support zones ($0.2\,l$ from each continuous edge) is converted to a top-steel requirement and checked against the top bars provided (EC2 9.3.1.3).
- Strip-model results (the older 1-D method, with the effective width b_e) are always printed alongside, as are the Table 3 / Table 8 coefficient values, with the FE span moment given as a percentage of the coefficient value.

8.3.3 UK NA 5.1.3(1)P option (c) and load arrangements

For slabs, the UK NA allows the **all-spans-loaded** arrangement alone, with 20 % redistribution of support moments (hogging $\times 0.8$, sagging increased by 0.2 times the mean hogging moment), when all three conditions hold:

Condition	Rule
Variable to permanent	Q_k / G_k not more than 1.25
Imposed load	Q_k not more than 5 kN/m², excluding partitions
One-way bay area	More than 30 m² (span × panel width). Not applied to two-way panels

If the slab qualifies and "20 % support redistribution" is on, option (c) is used and the working says so. If the slab does not qualify, or the switch is off, the FE falls back to arrangements (a)/(b): a continuous run is solved with alternating and adjacent spans loaded and **enveloped**; a single panel with continuous edges gets a checkerboard solve in addition to the all-loaded one (continuous edges released to simple supports under a $\pm q/2$ pattern). The working lists the arrangements and which governed each design value. A 2-span run is flagged because Table 3 is stated for three or more spans.

8.3.4 Moment redistribution and K'

The redistribution ratio δ describes the moments designed for. Table 3, Table 8 and the FE with option (c) all carry 20 % at supports, so δ used is the smaller of the entered value and **0.80** for any slab with hogging that uses them. Only an elastic FE run, or a slab with no hogging, leaves the entered δ in charge. A smaller entered δ is kept (conservative). $K' = 0.598\,\delta - 0.18\,\delta^2 - 0.21$ (UK NA 5.5(4), $k_1 = 0.4$, $k_2 = 1.0$ up to C50/60), optionally capped at 0.168.

8.4 Design checks

Every check produces a utilisation. The slab passes only if all pass. The status is WARN if everything passes but the worst utilisation is above 95 % or the working carries warnings. The ranking includes: Moment (each face and direction), Steel area, Deflection l/d, Deflection long-term, Shear (each support), Punching (each point load), and the detailing rows below.

8.4.1 Flexure (EC2 6.1)

Per metre width, for each location (span, supports, each direction):

- $K = M/(bd^2 f_{ck})$, $b = 1000$ mm.
- $z = (d/2)\left[1 + \sqrt{1 - 3.53K}\right]$, limited to $0.95d$; $x = 2.5(d - z)$; $A_s = M/(f_{yd} z)$, $f_{yd} = f_{yk}/1.15$.
- K above K': a **doubly reinforced** section is needed, which this node does not design. The location fails (utilisation shown as 1.25 on the ranking).
- $A_{s,min} = \max(0.26 f_{ctm}/f_{yk},\ 0.0013)\,bd$, applied where the moment is not zero; $A_{s,max} = 0.04bh$. $f_{ctm} = 0.30 f_{ck}^{2/3}$.
- The moment resistance of the bars provided uses $a = A_s f_{yd}/(0.567 f_{ck} b)$, $M_{Rd} = A_s f_{yd}(d - a/2)$. Utilisation is the larger of M/M_{Rd} and $A_{s,req}/A_{s,prov}$.
- d is measured to the bars of each layer with the cover of that face (bottom and top covers are separate inputs).

8.4.2 Shear (EC2 6.2.2)

- V_{Ed} at d from the support face: $V_{Ed,d} = V_{Ed} - nd$.
- $v_{Ed} = V_{Ed,d}/(bd)$; $k = 1 + \sqrt{200/d}$ up to 2.0; $\rho_l = A_{sl}/(bd)$ up to 0.02, with A_{sl} the tension steel actually at that section.
- $v_{Rd,c} = 0.12\,k\,(100\rho_l f_{ck})^{1/3}$, not less than $0.035\,k^{1.5}\sqrt{f_{ck}}$.
- There is one check per support or edge, each with its own steel: a simple support uses the anchored share of the bottom bars (default 50 %); a continuous support uses the top bars.
- The slab is checked against $v_{Rd,c}$ only; **no shear reinforcement is designed in a one-way or two-way slab**. If v_{Ed} exceeds $v_{Rd,c}$ the check fails and you increase depth or steel.

8.4.3 Punching under point loads (EC2 6.4)

Run for each active point load unless "Punching check" is off.

- Loaded area: a square of side b. Loaded perimeter $u_0 = 4b$ for an internal load; at an edge or corner within d, EC2 6.4.5(3) values ($u_0 = b + 3d$ not more than $3b$ at an edge; $3d$ not more than $2b$ at a corner).
- Basic control perimeter u_1 at $2d$ from the loaded area, $u_1 = 4b + 4\pi d$, clipped where it runs off the slab; near an edge or corner the shorter Figure 6.15 perimeter applies (6.4.2(4)).
- β: 1.15 internal, 1.4 edge, 1.5 corner (Figure 6.21N, UK NA 6.4.3(6)).
- $v_{Ed} = \beta V_{Ed}/(ud)$ against $v_{Rd,max} = 0.5\,\nu f_{cd}$ at the loaded face and $v_{Rd,c}$ at u_1, with $\rho_l = \sqrt{\rho_{lx}\rho_{ly}}$ from the **provided** bottom steel and d the mean effective depth.
- When a load is within d of an edge the working notes that edge reinforcement to EC2 9.3.1.4 is required.
- If the check fails the load is reported ("Punching reinforcement, a larger bearing area or a deeper slab is required"). The node does not design punching links for point loads.

8.4.4 Deflection (EC2 7.4)

Two checks, both in the verdict.

1. **Span/effective depth (7.4.2, Exp. 7.16a/b; UK NA Table NA.5).** Allowable $l/d = N \times K \times F_3$ (capped at $40K$), then $\times F_1 \times F_2$.
   - N from $\rho = A_{s,req}/(bd)$ against $\rho_0 = \sqrt{f_{ck}}/1000$. Compression steel is taken as zero.
   - K from the support condition: simply supported 1.0, end span 1.3, interior 1.5; for a two-way panel K follows the number of continuous long edges (two: 1.5; one: 1.3; none: 1.0) and the check is on the short span.
   - $F_3 = 310/\sigma_s$ up to 1.5, with $\sigma_s = f_{yd}(A_{s,req}/A_{s,prov}) \times (\text{characteristic SLS load}/\text{ULS load})$, after UK NA Table NA.5 Note 5. Point and line loads enter as equivalent UDLs.
   - F_1 and F_2 are 1.0; $F_2 = 7/\text{span}$ only with "Brittle partitions" and span over 7 m. The working lists the cap check (Note 6) when it applies.
2. **Explicit long-term deflection (7.4.3 and Annex B).** The quasi-permanent elastic deflection (from the plate when the FE is on) is scaled by the ratio of gross to effective stiffness. Creep $\varphi(\infty, t_0)$ comes from RH, age at loading, h_0 and the cement class; $E_{cm}/(1+\varphi)$ gives the long-term modulus. Shrinkage strain $\varepsilon_{cs} = \varepsilon_{cd} + \varepsilon_{ca}$ (3.1.4). The section is cracked where M_{qp} exceeds M_{cr} ($\zeta = 1 - 0.5(M_{cr}/M_{qp})^2$, Exp. 7.19), curvatures are interpolated between the cracked and uncracked states, and the shrinkage curvature of Exp. 7.21 is added. The long-term cracked deflection is checked against **span/250** (EC2 7.4.1(4)); span/500 is also calculated. The results panel and the working show the uncracked, cracked and shrinkage components.

8.4.5 Crack control (EC2 7.3.3)

For a slab in bending in a building with h up to **200 mm**, no specific measures are needed (7.3.3(1)) and the node says so. Deeper slabs are checked on every layer against Table 7.2N (bar size, corrected by Exp. 7.6N) **or** Table 7.3N (spacing), $w_k = 0.3$ mm (UK NA Table NA.4), with the steel stress under the quasi-permanent combination, $\sigma_s = f_{yd}(A_{s,flex}/A_{s,prov})(\text{quasi-permanent}/\text{ULS})$ and divided by δ for hogging steel only.

8.4.6 Detailing, cover, fire, anchorage

- **Bar spacing (9.3.1.1(3)).** Principal $2h$ not more than 250 mm; secondary $3h$ not more than 400 mm. These are the tighter limits for areas of maximum moment and concentrated loads, applied to every principal layer (both bottom directions and the top bars).
- **Secondary steel.** At least 20 % of the principal steel **provided** (9.3.1.1(2)). If the FE gives a transverse moment M_y, the steel to resist it is also calculated and the larger governs. $A_{s,min}$ is not added to M_y, because 9.3.1.1(1) applies it to the main direction.
- **Nominal cover (4.4.1, UK NA, BS 8500-1 Table A.4).** c_{nom} is the larger of the bond cover (bar diameter, at least 10 mm) plus Δc_{dev} of 10 mm, and the durability cover from BS 8500-1 Table A.4 and EC2 Table 4.4N (at the Table 4.3N

class for the working life, with the -1 class for slab geometry). Each face is checked against its own bars. Status OK / **Review required** (bond met, durability cover or concrete class below the BS 8500 requirement; does not fail the slab) / FAIL (bond minimum not met). In automatic mode the covers are set to the minimum (bottom also covering the fire axis distance, rounded up to 5 mm), so the check passes by construction and does not stand as the governing check. BS 8500-1 also sets a lowest concrete class for the exposure; it is checked and reported.

- **Fire (EN 1992-1-2).** The rating's minimum thickness h_s and axis distance a are compared with h and $a = c_{nom} + \phi/2$ of the bottom bars. Values used (h_s / a in mm):

Rating	One-way (Table 5.8, column 3)	Two-way, l_y/l_x up to 1.5 (column 4)	Two-way, 1.5 to 2 (column 5)
REI 60	80 / 20	80 / 10	80 / 15
REI 90	100 / 30	100 / 15	100 / 20
REI 120	120 / 40	120 / 20	120 / 25
REI 240	175 / 65	175 / 40	175 / 50

A two-way panel above l_y/l_x of 2 uses the one-way column. Simply supported values are used for continuous slabs too; they are the more onerous, and the 5.7.3 reductions are not applied.

- **Anchorage and laps (8.4, 8.7.3).** l_{bd} and l_0 per bar layer from $f_{bd} = 2.25\,\eta_1\eta_2 f_{ctd}$, with the bond condition from h, a_2 from cover and spacing, and α_6 from the lapped percentage; both the design-stress length and the full-f_{yd} length are reported.

8.5 Optimiser

"Find optimised sections" tries depths in 25 mm steps from the fire minimum (not below 100 mm) up to a little beyond the current depth, and for each depth the lightest passing bar arrangement from a fixed list (H10 to H25 at 125 to 200 mm spacing). It ranks the results by A1-A3 embodied carbon per m² (concrete factor $\times$ depth plus an estimated steel mass $\times$ rebar factor). Nothing is searched until the button is pressed. The results stay while the inputs are unchanged. Search results are kept while the inputs are unchanged. Flat and ribbed types are not searched.

8.6 Outputs and results panels

- **Drawing.** Plan with supports and bar layout to scale; section A-A with depth, cover, bars and top steel over supports; diagrams of bending moment, shear force, deflection and required A_s, cut through the solved plate (axis and position selectable) or taken on a 1 m strip when the FE is off.
- **3D view.** The panel with loads (area, point, line), the bars, and an optional **heat map** of $A_{s,req}$ on the bottom or top face (envelope or per direction). Loads are draggable.
- **Working panels.** Actions and combinations; analysis (the FE rows above, with the Table 3 or Table 8 comparison); flexure per location; deflection (l/d factors, creep and shrinkage, cracked/uncracked); shear per support; punching; crack control; cover; anchorage; and a code-compliance register.
- **Node card.** Lists each design with its slab type, governing check and utilisation.

8.7 Reports

The slab's calculation note is built by the shared module-report builder, so it follows the same layout and options as the Column, Beam and other modules (Chapter 14 describes the window and formats; it re-reads the live design each time). Contents:

- cover with PASS / FAIL and the governing check; summary of checks;
- figures: slab plan, section A-A, the design diagrams (or the whole drawing), the 3D view and, where the FE is on, bottom and top reinforcement heat maps;
- the slab model (type, geometry, analysis basis, loads in the calculation, materials, cover and fire, anchorage);
- warnings and notes, exactly as the module shows them;

- every working section with substituted numbers (the concise summaries are omitted so nothing prints twice);
- optimised sections, when you ran that search from the module;
- quantity and carbon with charts (stage split, material split, concrete and rebar source comparisons), and references.

8.8 Quantities and embodied carbon

Item	Default
Quantity (identical panels)	1
Volume	$h \times L \times L_y \times$ quantity (area is the panel, not the loaded width)
Concrete mass	volume $\times$ 2400 kg/m³ (plain concrete; reinforcement is its own line)
Concrete factor	CEM II/A-L, 217 $kgCO_2e/m^3$
Rebar	Allowance of 110 kg/m³ of concrete (this is the take-off, not the designed bar layout)
Rebar factor	IStructE Table 2.5 "UK reinforcement bar (consumption average)", 0.72 $kgCO_2e/kg$
A4 / A5	12 % / 4 % of A1-A3

The slab uses the **allowance rate**, not the bars in the layout. The same bar-bending-schedule override as in the Beam (Chapter 6, 6.8) lets you enter your own kg/m³ rate (with element-type presets) or a total in kg per panel. Carbon scope can be the current slab or all slabs on the node.

8.9 Connections to other nodes

8.9.1 Into a slab

Source	What arrives
Loads (load set)	Only **area** loads (kN/m²) are accepted. Rows in kN or kN/m are rejected with the reason. The wire chooses which slab field a class drives: screed, finishes and services, movable partitions or imposed q_k (defaults: permanent to finishes, variable to q_k). Ticking several fields makes one value cover them (it is written to the first and the rest are zeroed, so the load is not counted twice). Sending a permanent load to partitions classes it as variable, and the wire warns
Math	A result of unit kN/m² (for example a Loads row $\times$ a factor). A result with another or unknown unit is rejected
IFC	The designed element's geometry and loads

Rules: where two wires drive the same field the last synced wins (and the wire menu warns). Deleting a wire releases the fields it was filling (they are set to 0 and the preset to custom). Wind and Sketch-ET wires are accepted by the map, but the sync writes no loads from them into a slab.

8.9.2 Out of a slab

Each design has an OUT dot "Slab area loads gk / qk (incl. self-weight)". It carries the **characteristic** area loads in kN/m²: g_k (self-weight, screed, finishes) and q_k (imposed and partitions), after any muting.

- To a **Beam**: converted to line loads through a loaded width (Chapter 6, 6.9.1). A one-way slab gives a uniform band of area load $\times$ width, with the width defaulting to $l_x/2$. A two-way slab sheds at $45°$: a trapezoid on a beam along the long edge, a triangle on a beam along the short edge, each peaking at area load $\times$ width. A tick can set the beam span from the slab edge length. The slab's ψ category is copied to the beam. Width, edge choice and muting are controlled on the beam.
- To a **Column**: a direct quarter-panel tributary area ($l_x/2 \times l_y/2$) on the characteristic area loads.
- To **Math**: as DL and LL values (g_k and q_k).

The slab's point and line loads are not sent to a beam; only the area loads are.

8.10 Tips, limitations and known caveats

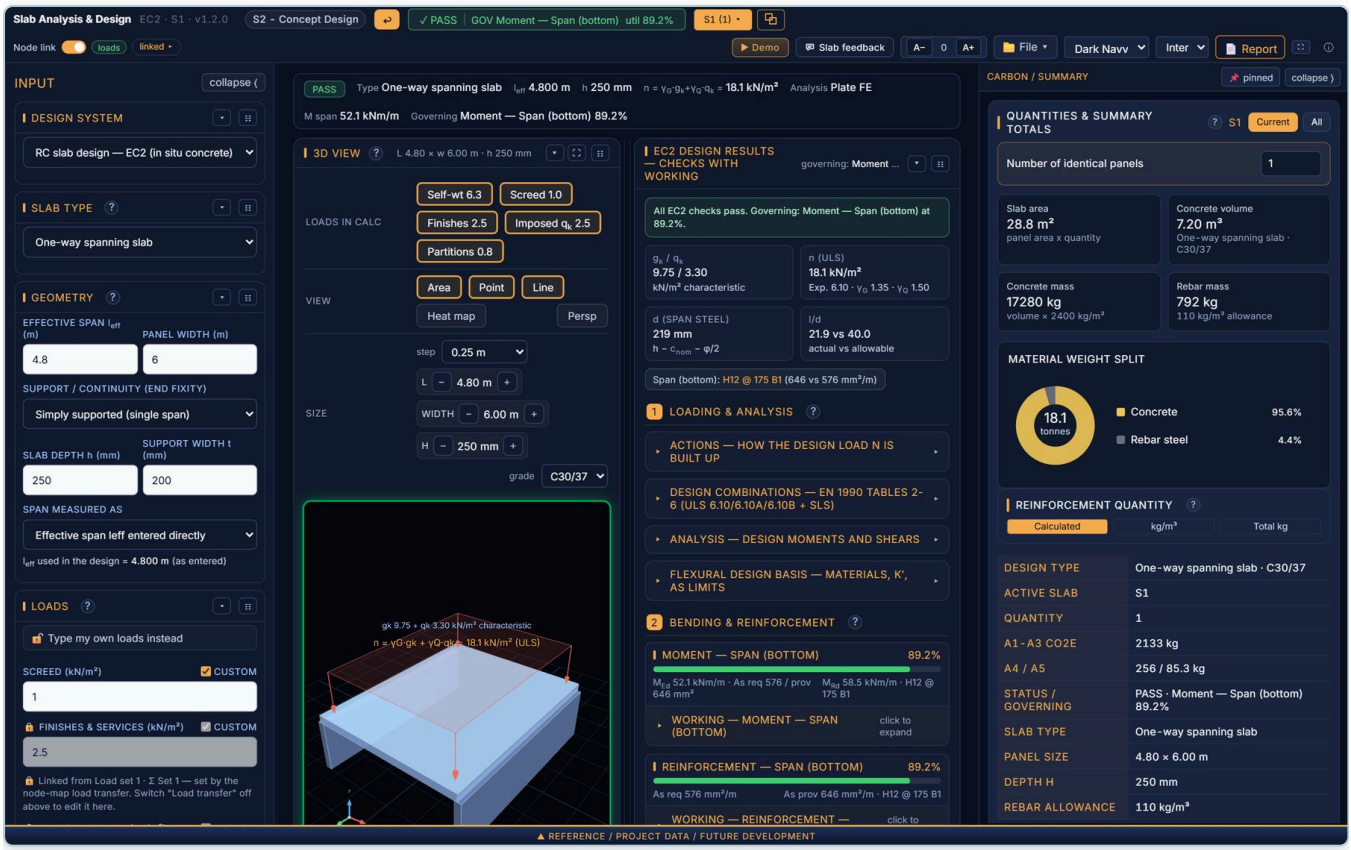
- **Defaults are placeholders, not a design.** The unmodified default slab (4.8 m simply supported, h 175, H12 at 175, C30/37, office loading) does not pass: the l/d check is about 1.4 times its limit and the long-term deflection about 1.7 times span/250. Moment utilisation is about 97 %. Change depth or bars.
- **Flat slab, ribbed slab and ComFlor are locked.** Do not rely on any result for them.
- **No shear reinforcement and no punching links** are designed; these checks only pass or fail.
- **Doubly reinforced** sections are not designed. K above K' fails the location.
- **Check the option (c) conditions.** If the slab does not qualify, the node switches to load-arrangement enveloping (FE) or fails the coefficient basis. Do not ignore the message.
- **FE is uncracked and elastic** for the moments (Poisson's ratio 0.2). Cracking enters only the deflection.
- **Concentrated loads depend on the smoothing band.** Look at the smoothing warning; set the band by hand if you disagree with the automatic width.
- **Mesh density** costs a great deal: the work grows steeply with the number of nodes. Normal is converged for design; go coarse for fast load dragging and fine only for small loaded areas or loads close to a support.
- **Muted loads stay muted downstream.** They are excluded from what the slab sends to beams and columns.
- **Adjacent spans.** The continuous models assume similar, equally loaded spans; enter the difference to be warned.
- **The beam's creep coefficient is fixed**; the slab's follows RH and age.
- **Results are kept.** A change that does not alter a number does not redesign the slab, and the optimiser runs only when asked.

> **References.** The standards, guidance and data sources used in this chapter are listed, with web links, in Chapter 17.

Screen reference - panel by panel

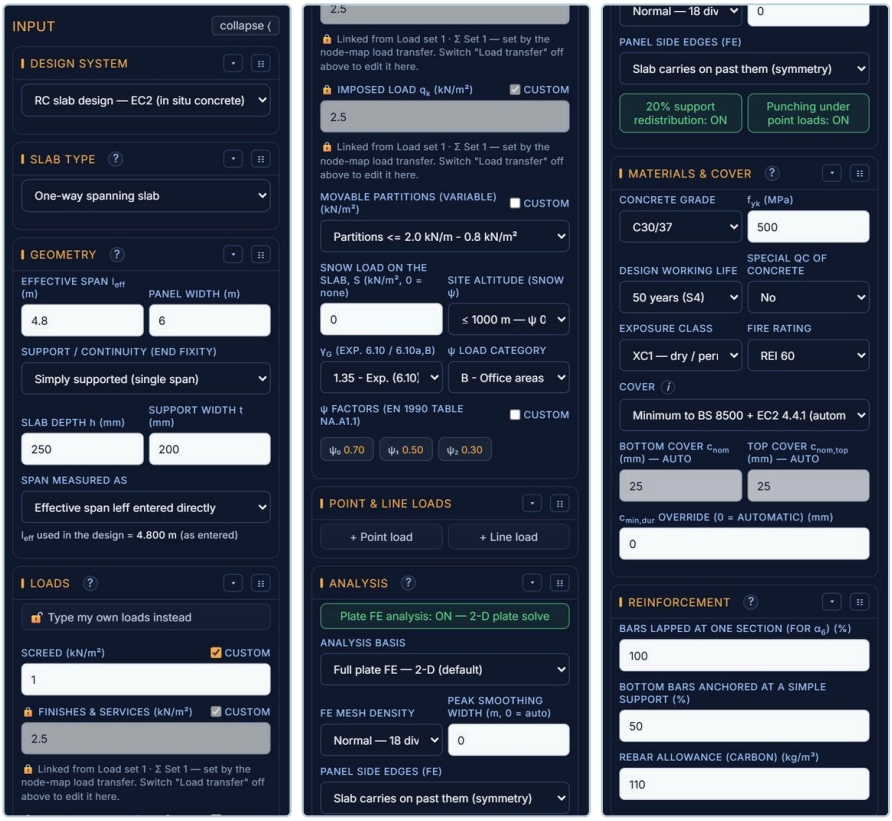
These screenshots show every panel of the module exactly as it appears on screen, scrolled from top to bottom, so you can match any field, button or result to the explanations in this chapter. Cards that are collapsed by default are shown expanded.

Slab: the whole screen



Slab: the whole screen.

Slab: input panel (left of the screen)

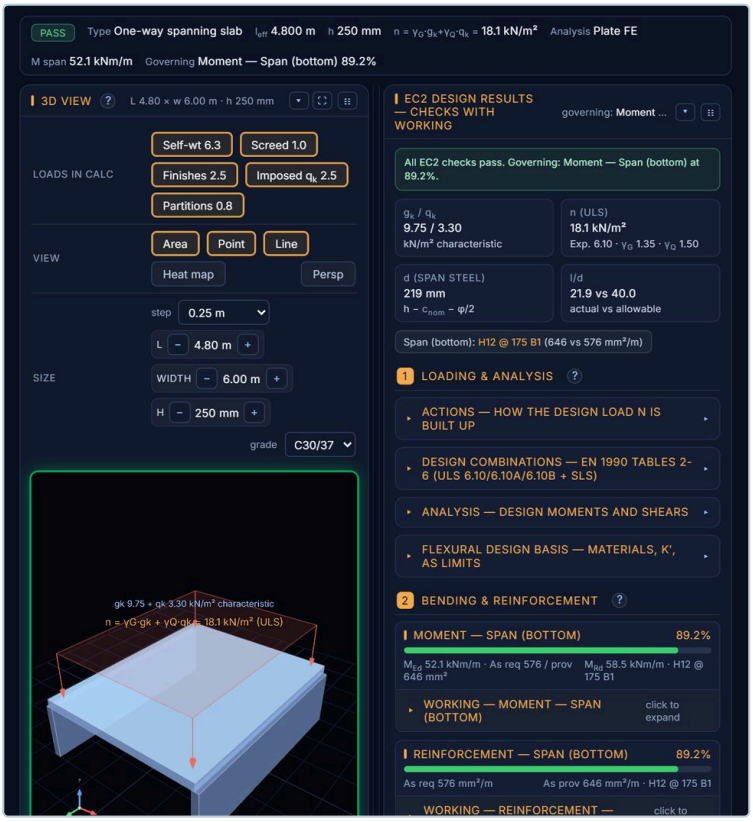


Slab, input panel (left of the screen), parts 1-3 of 4. Cards shown: Design system, Slab type, Geometry, Loads, Point & line loads, Analysis, Materials & cover, Reinforcement.

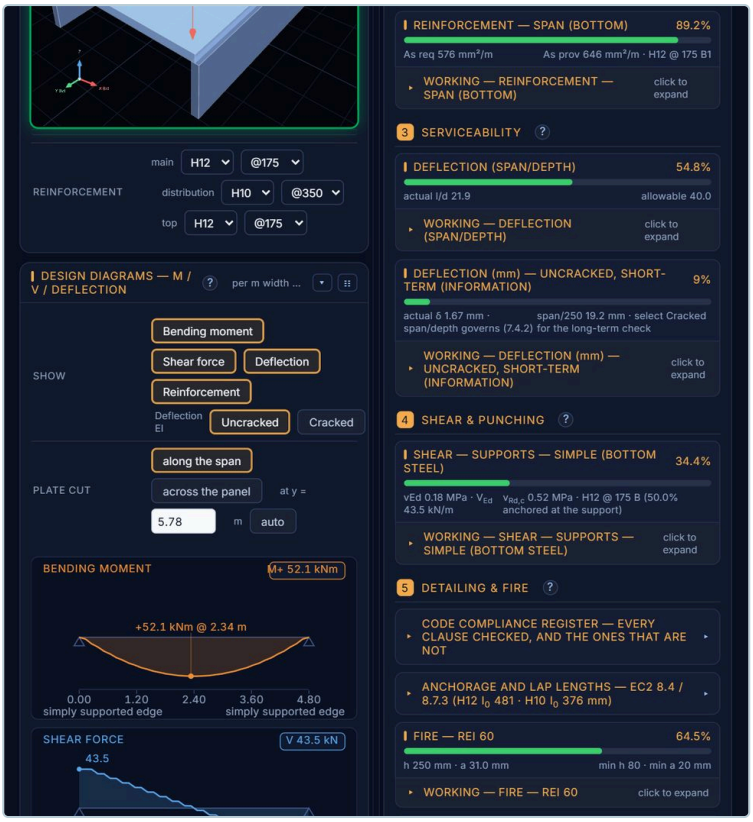


Slab, input panel (left of the screen), part 4 of 4. Cards shown: Reinforcement, Design options.

Slab: working panel (centre of the screen)



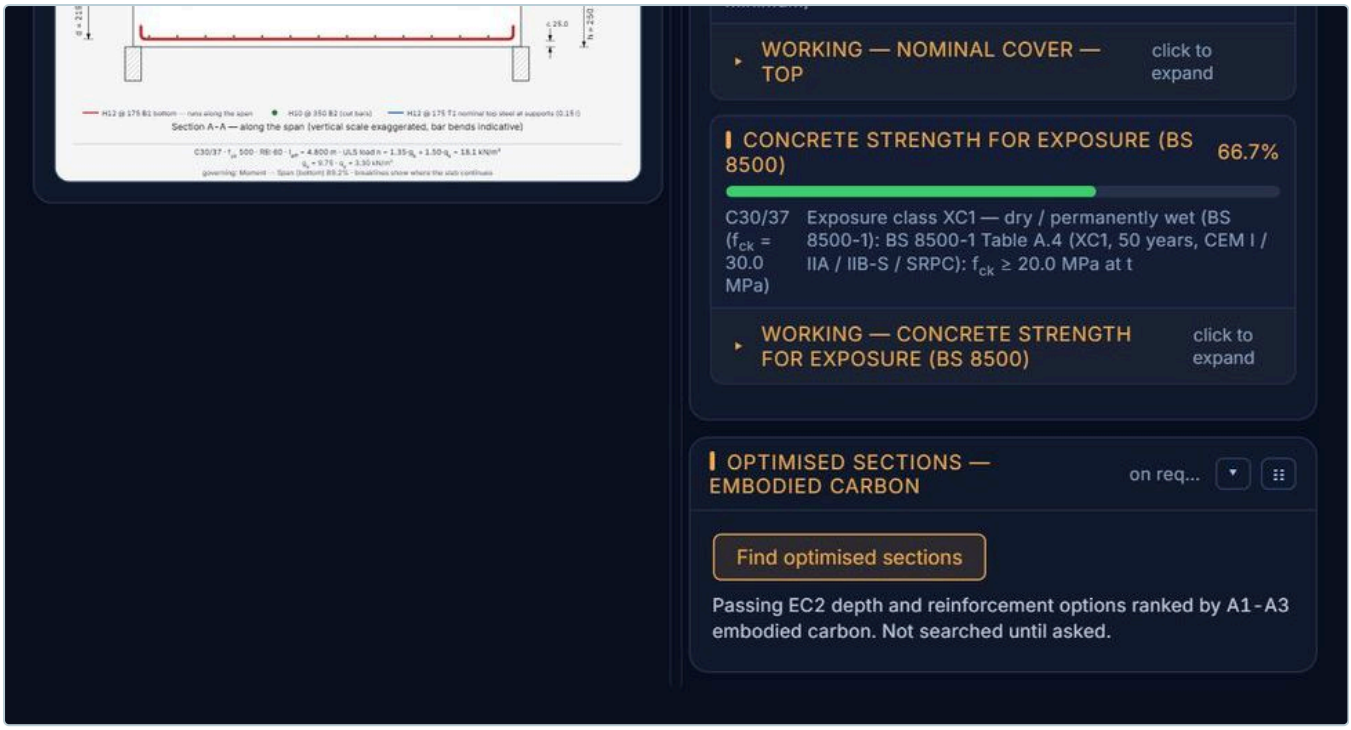
Slab, working panel (centre of the screen), part 1 of 4. Cards shown: 3D view, EC2 design results — checks with working.



Slab, working panel (centre of the screen), part 2 of 4. Cards shown: Design diagrams — m / v / deflection, Reinforcement — span (bottom), Working — reinforcement — span (bottom), 3 serviceability, Deflection (span/depth), Working — deflection (span/depth), 4 shear & punching.

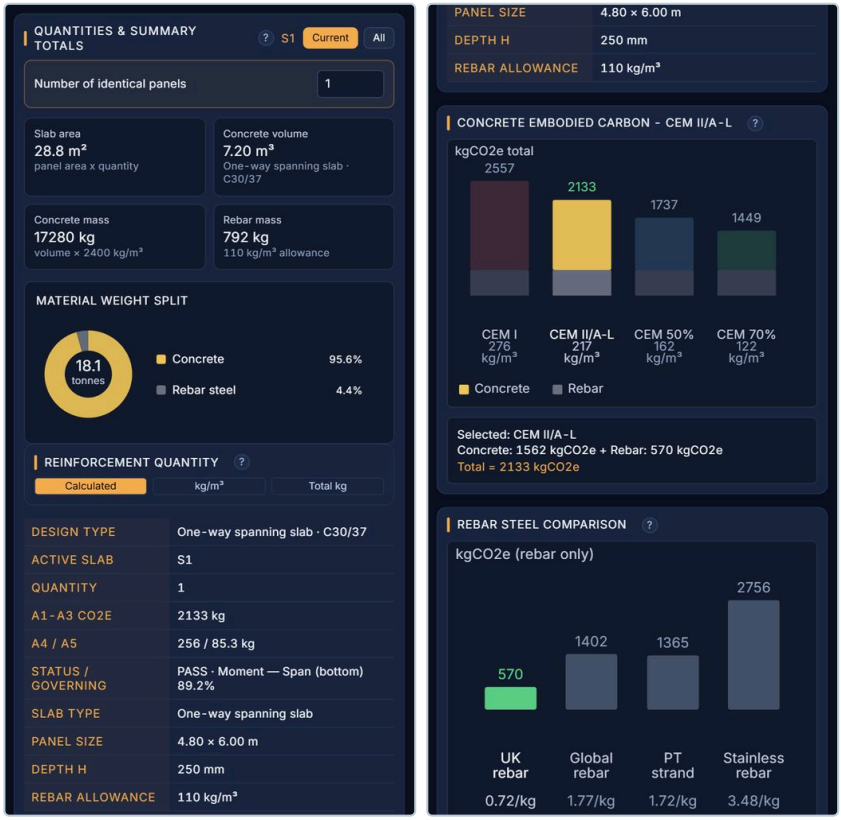


Slab, working panel (centre of the screen), part 3 of 4. Cards shown: Live 2D drawing, Working — fire — REI 60, Secondary transverse steel, Working — secondary transverse steel, Max principal bar spacing, Working — max principal bar spacing, Max secondary bar spacing, Working — max secondary bar spacing.

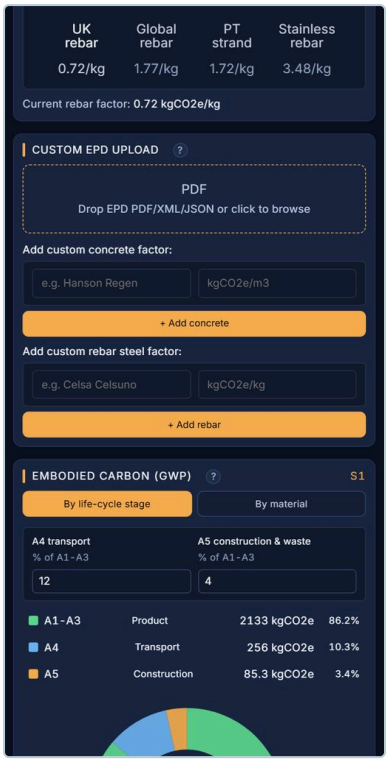


Slab, working panel (centre of the screen), part 4 of 4. Cards shown: Live 2D drawing, Working — max secondary bar spacing, Crack control — span (bottom), Working — crack control — span (bottom), Nominal cover — bottom, Working — nominal cover — bottom, Nominal cover — top, Working — nominal cover — top.

Slab: results panel (right of the screen)



Slab, results panel (right of the screen), parts 1-2 of 4. Cards shown: Reinforcement quantity, Concrete embodied carbon - CEM II/A-L, Rebar steel comparison.





Slab, results panel (right of the screen), parts 3-4 of 4. Cards shown: Custom EPD upload, Embodied carbon (GWP) S1.

CHAPTER 9

Wind designer

The Wind designer works out wind actions on a UK building (or a few simple non-building structures) to **BS EN 1991-1-4:2005+A1:2010 with the UK National Annex (NA)**. You locate the site, describe the building, and the node returns the peak velocity pressure, the pressure on every wall and roof zone, the internal pressure, the overall horizontal force for each wind direction and the worst local suctions. It produces a Word-style report, and it can send its governing horizontal force to the Beam, Slab, Column, Foundation and Base plate nodes.

The node designs to the UK Annex only. A site outside the area covered by NA Figure NA.1 is flagged "OUT OF BOUNDS" and the numbers shown are not valid.

9.1 What it does

For each wind direction the node:

1. Reads the basic wind speed $v_{b,map}$ from the digitised NA Figure NA.1 at the site (or takes a value you type).
2. Applies the altitude, directional, season and probability factors to get v_b, then q_b.
3. Reads the exposure factor $c_e(z)$ (NA Figure NA.7) from the distance upwind to the sea, and the town factor $c_{e,T}$ (NA Figure NA.8) if the site is in a town. This gives the peak velocity pressure $q_p(z)$.
4. Builds the pressure zones on the walls and roof (EN 1991-1-4 Section 7.2, UK NA Tables NA.4 to NA.8) for the building geometry, with optional funnelling on chosen walls.
5. Works out the internal pressure coefficient c_{pi}.
6. Sums the zone forces into an overall horizontal force F_w (with the correlation reduction, the structural factor $c_s c_d$ and optional friction), and finds the worst net local pressure.
7. Shows all of it as a site map, a plan with zones, a 3D model with wind flow, calculation sheets, comparison and chart tabs, and a report.

Everything is recalculated live whenever an input changes.

9.2 Opening the node

- On the node map, add a **Wind designer** node (tag "EC1-1-4 - UK NA"). Double-click the node card to open the full-screen module.
- If you open the Wind module with no wind node in the project it offers **+ Add wind case**.
- The module header has a node switcher if the project holds several wind designs, and an "Add wind" option.
- The node card has **no OUT socket** of its own; it exposes a per-design output dot (see 9.10).

The module has an **Inputs** column on the left (a stack of collapsible cards that you can drag to reorder; the column can be collapsed to a rail) and a stage on the right with seven tabs.

Tab	What it shows
Overview	The site map and the 3D model side by side. Includes the Google aerial and Google photorealistic 3D options.
Wind map	The NA Figure NA.1 map: coastline, isopleths, shaded $v_{b,map}$ field, OS grid, site marker, shore rays; optional Google aerial/3D map.
Plan - zones	The scale plan with zone boxes, dimensions, wind arrows, a zone table, outline drawing, PDF/image underlay.
3D and flow	3D model with zones, traffic-light colours, force boxes, streamlines, dimension boxes, funnelling tags and a north dial.

Tab	What it shows
Results	The numbered calculation sheets for one direction at a time (1 to 8, below).
Compare	Normal values against funnelling values, zone by zone.
Charts	The NA/EN charts redrawn from the data the design uses, with this design's point marked.

Each viewer (map, plan, 3D) has its own **Cards** switch to hide the floating cards and tool bars; the choice is remembered per viewer in the browser.

9.3 Inputs

The cards are listed here in the order of the office hand calculation, which is also the order they appear in. Every card has a **?** button that explains only that card's inputs, in order. Numbers use a decimal comma in the interface text; this chapter writes decimal points.

9.3.1 Site and wind climate card

Input	Units / options	Default	Notes
Site search	text	empty	Place name, OS grid reference (for example `TQ 3003 8039`), `lat, lon`, and (with the online lookup ticked) addresses and postcodes. A site found by search is **locked**.
Site lock	Lock / Unlock	unlocked	A locked site cannot be moved by a stray map click or drag. Rotation still works.
Look up addresses, postcodes and ground level online	tick	off	Sends the typed text to the lookup and the site position to an elevation service. See 9.3.1.1.
Wind climate	From the map (Figure NA.1) / Enter $v_{b,map}$ directly	From the map	In manual mode the site position still places the building on the maps and sets the fetch and directions.
$v_{b,map}$ (manual mode only)	m/s	none	Basic wind speed before the altitude factor. UK range is about 21.5 to 31 m/s.
Easting, Northing	km, OS National Grid	530, 181	Held to the metre. Also shown as a 10-figure grid reference with latitude/longitude, address and postcode in the grid reference card.
Altitude A	m above sea level	20	Filled automatically when the online lookup is on; you can overwrite it.
Simple altitude factor	tick	off	Uses $c_{alt} = 1 + 0.001A$ at every height (NA.2a), conservative. Off uses NA.2b above 10 m.

If the marker is off the coastline traced from Figure NA.1 the card says so and a warning appears above the stage.

9.3.1.1 Online lookups and privacy

Nothing is fetched from a tile server for the wind map itself. The following only happen when you tick the online option (and for the Google views, when you switch them on):

- Postcodes go to postcodes.io. Addresses are suggested from Google Places when a Google key is connected, otherwise from Photon/OpenStreetMap.
- The site position goes to the Google Elevation API, or Open-Meteo as the fallback, to fill altitude A.

9.3.2 Plan outline card

Input	Units / options	Default	Notes
Roof	Flat / Monopitch / Duopitch / Hipped	Flat	Sets the roof zone tables (Table NA.5, NA.6, NA.7, NA.8). Choosing a pitched roof with no pitch sets $15°$.
Roof detail	see 9.3.2.1		Appears under the roof type.
Orientation	degrees clockwise from north to the building $+y$ axis	0	Decides which wall faces each wind, hence c_{dir} and the fetch for each face.
Plan shape	Rectangle / Square / Drawn polygon	Rectangle	
Plan size X, Y	m	20 by 12	Typed on the 3D view's dimension lines (click the dimension box).
Height	m, to eaves	12	Also typed in the 3D view. For a flat roof this is roof level; for a monopitch the low eave.
Draw plan outline	button		See 9.4.
Release the IFC lock	button		Only shown after an IFC import.

9.3.2.1 Roof detail

The roof belongs to the building, not to the wind. The height you enter is the **eaves** height; the **top** (ridge or high eave) is derived and is the reference height h for zones, z_e and $e = \min(b, 2h)$.

Roof	Extra inputs
Monopitch	High eave on (which wall carries the high eave), pitch α. Wind onto the low eave is $\theta = 0°$, onto the high eave $180°$, along the eaves $90°$ (Figure 7.7).
Duopitch	Ridge runs along x or y, pitch α (typical values include $-15°$ for a valley). Figure 7.8.
Hipped	Ridge direction, main pitch α_0, hip-end pitch α_{90} (blank = same as main). The windward face pitch governs (NOTE 3 to Table NA.8). The hipped zones are true polygons (Figure 7.9).
Flat	Eaves detail: Sharp / With parapet / Curved eaves / Mansard eaves. Parapet height h_p (default 0), curved eaves radius ratio r/e, or mansard angle. A parapet references the roof to $h + h_p$ (7.2.3 (3)).

Pitches below $5°$ in absolute value use the flat-roof table. The pitched-roof tables run from $-45°$ to $+75°$; the node does not interpolate across the $+5°/-5°$ break and warns when the pitch is under $5°$.

9.3.3 Wind directions and factors card

Input	Options	Default	Notes
Season factor c_{season}	Table NA.2 values (year 1.00, winter 1.00, summer 0.84, or a single month)	1.0	Use below 1.0 only for temporary works in a known season.
Annual probability of exceedence	0.02 / 0.01 / 0.04 / 0.1 or typed	0.02	Gives c_{prob} from Expression 4.2 with $K = 0.2$, $n = 0.5$ (NA.2.8). 0.02 gives 1.0.
Take $c_{dir} = 1.0$ for every direction	tick	off	NOTE 4 to Table NA.1. Conservative; q_p becomes one number.
Add the four diagonal directions	tick	off	Adds $45°$, $135°$, $225°$, $315°$ relative to the building, using the c_{pe} of the nearest square-on case.

The wind directions themselves are not typed. They are the **normals to the building's faces**, at orientation $+k \times 90°$ (plus $45°$ diagonals if ticked). Each orthogonal case takes the maximum c_{dir} within $\pm 45°$ of its face normal (NOTE 3 to Table NA.1). A rotated building is therefore analysed square-on to each of its walls, not on fixed N/E/S/W bearings.

9.3.4 Terrain and exposure card

Input	Units / options	Default	Notes
Terrain	Country / Town	Country	Country: $c_{e,T} = 1.0$ (conservative). Town: $c_{e,T}$ from Figure NA.8.
Distance inside the town boundary	km (0.1 to 20)	2	Town only. Held in each direction to the distance to the shore, since a town cannot start beyond the sea. Taking the town edge (0.1 km) is the conservative reading.
Height of surroundings h_{ave}	m	0	Town only; with spacing x gives the displacement height h_{dis} (Annex A.5).
Spacing of obstructions x	m	0	Town only.
Measure the fetch to the shoreline from the map	tick	on	Per-direction distance to the sea; see 9.5.4.
Distance upwind to the shoreline	km (0.1 to 100)	20	Used only when the tick above is off; one value for all directions.
Ignore tidal water narrower than	km	0.5	In the fold "Shore settings (only if needed)".
Shore no further than	km	none	Manual cap, in the same fold. Conservative.
Orography is significant (Figure NA.2)	tick	off	
Orography factor c_o	-	1.0	Annex A.3. Below 50 m, q_p is multiplied by $((c_o + 0.6)/1.6)^2$.

9.3.5 Funnelling and analysis options card

Input	Options	Default	Notes
Funnelling	ticked wall by wall in the 3D view	off	See 9.6.
Use the net pressure coefficient for overall loading	tick	off	NA.2.27 f). One net coefficient replaces D + E for the overall force. Cladding still uses Table NA.4.
Surface friction	None / Smooth (c_{fr} 0.01) / Rough (0.02) / Very rough (0.04)	None	Table 7.10. Ignored automatically where 5.3 (4) allows (parallel area not more than 4 times the windward plus leeward area).
Air density	kg/m³	1.226	NA.2.18.
Strip height on the windward wall	m	blank	Blank is one strip of height b. A building taller than $2b$ may be split into strips (Figure 7.4).

9.3.6 Openings and internal pressure card

A switch at the top, **Include internal pressure** c_{pi}, turns internal pressure off ($c_{pi} = 0$). That is acceptable for the overall horizontal force on a closed building but not for cladding, fixings or roof uplift. The basis you had chosen is remembered when you switch it back on.

Basis	What it does
Unknown openings: +0.2 / -0.3	7.2.9 (6) NOTE 2. Both values are carried through; the worse governs. This is the default.
From the openings listed below	Each opening sits on a real wall (or the roof). Its role (windward, side, leeward) is worked out for each wind direction and c_{pi} follows 7.2.9 (3) to (6).
Openings spread evenly (enter μ)	Figure 7.13 with the opening ratio μ (default blank; range 0 to 1).
One dominant face (enter its c_{pe})	7.2.9 (5): $c_{pi} = 0.75$ to 0.90 times c_{pe} at the openings, interpolated on the area ratio (2 to 3 or more; default 2). Blank c_{pe} uses the windward D value.
Enter c_{pi} directly	A single value for every surface.

Each listed opening has: wall (named by its compass direction and $+x/-x/+y/-y$, or Roof), area (m², new openings default to 2 m²) and offset from the middle of the wall (m, positive towards the $+x/+y$ end, default 0; it decides side-wall zone A, B or C). Add openings with **+ Opening**, or click a wall on the plan and use **+ Opening here**. Adding an opening switches the basis to "From the openings listed". A small table shows, for each direction, which openings fall in which zone and the resulting c_{pi}.

9.3.7 Dynamic response cs cd card

Mode	Behaviour
Automatic (default)	$c_s c_d = 1.0$ where 6.2 (1) allows it. If not allowed it still uses 1.0 and shows a warning that this is not justified.
c_s from Table NA.3 $\times$ c_d	c_s from Table NA.3 (size and zone); c_d typed (1.00 to 1.25, from Figure NA.9, which is not reproduced). NA.2.20 NOTE 4 allows 1.0 for buildings up to 20 m.
Annex B detailed procedure	Needs construction type, fundamental frequency $n_{1,x}$ (blank = $46/h$, Annex F expression F.2, meant for multi-storey buildings over 50 m), damping δ_s (default 0.05), optional mass per length and c_f for aerodynamic damping.
Enter $c_s c_d$ directly	A typed product (default 1.0).

9.3.8 Other structures card

The **Structure type** defaults to Building (Section 7.2). Other types replace the zone calculation with a force coefficient or net coefficient:

Type	Basis
Tower, mast or chimney	7.6 to 7.9 with 7.13; section rectangular (Figure 7.23), sharp-edged ($c_{f,0} = 2.0$), circular (needs $c_{f,0}$ from Figure 7.28) or lattice (7.11). Inputs: breadth b, depth d, length l, end conditions (Table NA.10), solidity φ, corner reduction ψ_r.
Signboard or billboard	7.4.3, $c_f = 1.80$ where $z_g > h/4$ or b/h not more than 1. Inputs: width, height, clear height to underside.
Free-standing wall or fence	7.4.1, Table 7.9, zones from the free end. Inputs: height, length, solidity, returns. Solidity below 0.8 is treated as a plane lattice.
Parapet on a building	7.4.1 with $z_e = h + h_p$.

This card also holds the IFC import (9.4.2).

9.4 Describing the building

9.4.1 Drawing an outline

Use **Draw plan outline** (plan view bar or Plan outline card):

- Click the first corner, then type a length and press Enter (or click) for each side. $90°$ snap is on; the **90 degrees** chip (or the key `0`) toggles it.
- Type `6<30` for 6 m at $30°$, or `@4,3` for an offset.
- Enter on an empty box, or a click on the first corner, closes the outline. Backspace undoes the last point. Esc cancels.
- Drawing switches the shape to a polygon. **Back to a rectangle** returns to a rectangle (clears the polygon).
- Orientation is set with the rotate control on the plan bar or the Orientation field.

A **PDF page or image underlay** can be placed under the plan to sketch over (Underlay chip). It has page, scale, offset, opacity and invert controls.

For a polygon plan, each wall edge gets its own zones and the roof zones are cut to the outline. EN 1991-1-4 7.2.2 tabulates c_{pe} for rectangular plans only, so the **enclosing rectangle for each wind direction** is used for b, d and h/d, and the node warns that the result must be reviewed.

9.4.2 Importing an IFC model

In the Other structures card: **Import IFC model** (plain `.ifc` STEP files only; `.ifczip`, `.ifcxml` and zip are refused) or **Take from an IFC node** if the project has an IFC node.

- Triangles from all elements are collected; only the largest cluster in plan is kept, so a site model with separate outbuildings does not distort the footprint.
- The plan angle is taken from the dominant member directions, not from a minimum-area rectangle.
- Pitched roofs enter as an eaves height with the ridge on the long side.
- The footprint and height are then **locked**: only the orientation can change, until you press **Release the IFC lock** (the footprint then becomes an editable outline).
- A ghost of the imported mesh can be shown over the zones in the 3D view (**IFC model** chip).

9.5 What the calculation does

9.5.1 Basic wind speed and peak velocity pressure

For each direction:

$$c_{alt} = 1 + 0.001A \quad (z \le 10\ \text{m or the simple option}) \qquad \text{or} \qquad c_{alt} = 1 + 0.001A\left(\frac{10}{z}\right)^{0.2} \quad \text{(NA.2b)}$$

$$v_{b,0} = v_{b,map} \times c_{alt} \quad \text{(Equation NA.1)}$$

$$v_b = v_{b,0} \times c_{dir} \times c_{season} \times c_{prob}$$

$$q_b = \frac{0.5\,\rho\,v_b^2}{1000}\ \text{kN/m}^2, \qquad \rho = 1.226\ \text{kg/m}^3$$

$$q_p(z) = c_e(z) \times c_{e,T} \times q_b \quad \text{(NA.3a / NA.3b)}$$

EN Expression (4.8) is not used (NA.2.17).

- $c_e(z)$ is read from Figure NA.7 at the distance to the sea (0.1 to 100 km) and $z - h_{dis}$ (2 to 200 m). $c_{e,T}$ is read from Figure NA.8 (0.1 to 20 km into the town, never above 1.0); it is 1.0 in country terrain.
- With significant orography and c_o not 1.0: NA.4a for z up to 50 m. Above 50 m NA.4b is used only if c_r and I_v are supplied; otherwise 4a is applied and noted.

- A building taller than 200 m raises a warning because the charts are clamped at 200 m.

The chart data (Figure NA.1 coast, isopleths and $v_{b,map}$ field; Figures NA.7 and NA.8; Table NA.3; EN Figures 7.13, 7.23, 7.36) were traced from the vector artwork of the published documents. The module and the report state this.

9.5.2 Directional factor

Table NA.1 is held as twelve values at 30° spacing and interpolated linearly. An orthogonal case takes the maximum within $\pm 45^\circ$ of the face normal. A case set up for one specific skew bearing takes c_{dir} at that bearing and c_{pe} of the nearest orthogonal case (EN 7.2.1 (2)); the orthogonal cases still run, so the envelope cannot fall.

9.5.3 Site position and OS grid

The site is held as an OS National Grid reference. Conversion to and from WGS84 uses the OSGB36 Helmert transform and is accurate to a few metres. The grid reference is held to the metre and shown as a 10-figure reference, so it can be compared with an external grid reference finder. A postcode point can differ by roughly 17 m from another geocoder's position.

9.5.4 Distance to the shore (fetch)

For each wind direction the fetch is the **shortest** distance to the shore within $\pm 45^\circ$ of the direction ($\pm 15^\circ$ for a single skew bearing). Sweeping a sector, rather than one line, stops a ray slipping past a nearby estuary (2.5° steps).

The shore comes from the **OS Boundary-Line mean high water line** built into the app, which follows tidal rivers and estuaries up-country:

- Great Britain only. Northern Ireland falls back to the coastline traced off Figure NA.1; the OS file does not cover it.
- Tidal water counts as "sea" only if its clear width along the ray is at least the **Ignore tidal water narrower than** setting (default 0.5 km), so a narrow tidal river in a city centre is not treated as the shore.
- The search runs to 400 km, but the chart is read at 100 km; fetches of 100 km or more are labelled "over 100 km (Fig. NA.7 limit)".
- The map draws each ray as a dashed line with a distance pill. Short rays on the national map are also drawn in a bottom-right close-up.
- OS attribution ("Contains OS data (c) Crown copyright and database right") is shown on the terrain card and on the map.

9.5.5 Walls

Wall pressure coefficients come from Table NA.4 interpolated linearly on h/d (zones A, B, C on the sides, D windward, E leeward). The zone widths follow Figure 7.5 using $e = \min(b, 2h)$ where b is the crosswind breadth. The windward wall D follows the Figure 7.4 reference-height profile: one strip of height b, or several strips if the strip-height option is used for a tall building. Side and leeward walls use $z_e = h$. h is the top of the building, including the high eave or ridge; for a flat roof with a parapet the reference is $h + h_p$.

9.5.6 Roofs

Flat roofs use Table NA.5 (zones F, G, H, I, with the eaves detail options); monopitch NA.6, duopitch NA.7 and hipped NA.8, each by wind angle $\theta = 0^\circ, 90^\circ$ (and 180° where the figure has it). The roof wind frame works out θ from the building orientation, ridge direction and wind bearing. In the 3D view, each roof zone is folded onto its roof plane.

9.5.7 Funnelling

NA.2.27 e) funnelling between neighbouring buildings is applied **wall by wall** and replaces the side-wall coefficients A, B, C of Table NA.4 (-1.2, -0.8, -0.5) with -1.6, -0.9, -0.9. D and E are unchanged. Tick it in the 3D view: each wall carries a clickable **Funnelling** tag; red means on. The face is named by its outward normal clockwise from the building

$+y$ axis ($0°$, $90°$, $180°$, $270°$). It affects a wall only in the wind cases where that wall is a **side** wall; a ticked wall that is windward or leeward in a given case keeps its normal values. Where only one of the two side walls funnels, the side zones split into separate left and right entries (areas are not halved). Polygon plans funnel on all faces if any is ticked.

The node warns that NA.2.27 d) 1) allows funnelling only where the gap to the facing building is between $e/4$ and e; confirm the gap or untick the wall.

9.5.8 Internal pressure

Handled as described in 9.3.6. Beyond the options listed, the node also supports the open-silo ($c_{pi} = -0.60$) and vented-tank (-0.40) values of 7.2.9 (8), used by the Other structures route. Internal pressure $w_i = q_p \times c_{pi}$, evaluated at the relevant z_e.

9.5.9 Overall force and the worst local suction

For each case the overall horizontal force is the windward plus leeward (D + E) force (or the net coefficient if selected), multiplied by the correlation factor of 7.2.2 (3) (1.0 for h/d not under 5, 0.85 for h/d not over 1, linear between; NA.2.19 extends this to the horizontal component from the roof), times $c_s c_d$, plus friction if applicable. The result F_w is reported per direction; the **governing horizontal** case is the largest $|F_w|$, and the **governing uplift** case is the one with the most negative total roof force. The **worst local** pressure is the most negative net pressure ($w_e - w_i$) over every direction, zone and c_{pi} value.

9.5.10 Warnings

The warning list above the stage reports, among other things: out-of-bounds site; marker off the traced coast; $v_{b,map}$ defaulted; polygon plan; pitch under $5°$; $c_s c_d$ not justified; c_d not set for the table mode; funnelling gap; fundamental frequency assumed; building over 200 m.

9.6 Views

9.6.1 Wind map

The map is Figure NA.1 itself, drawn from the traced vector data: the coastline, the eleven isopleths and the OS grid, with the shaded $v_{b,map}$ field. You can pan, zoom, click to place the site (unless locked), and see the shore rays for each direction. A north badge is always on screen. A Google aerial (2D) or Google photorealistic 3D view can be switched on; the aerial/3D views use a Google key connected to Civil ET and an online connection.

9.6.2 Plan - zones

Zones are drawn on the plan as traffic-light boxes (by size of the selected quantity), with labels placed at the zone centroid, with leader lines when displaced. Everything is laid out so that **nothing overlaps**: text, dimension lines, witness lines, force boxes and wind arrows. For rectangular plans dimension chains are drawn in the frame of the case shown.

- **Wind from**: choose the direction (bearing).
- **Metric** drop-down: w_e (kN/m^2), $w_{e,fun}$, net, c_{pe}, force (kN), area (m^2) or length (m).
- Chips: Zones, Roof, Draw, 90 degrees, Dims (zone dimensions), Underlay, rotate control, Table, and a pan/zoom reset.
- Click a wall to open a pick card with its zones and **+ Opening here**.
- The table under the drawing lists zones and pressures for every direction.

9.6.3 3D and flow

- Orbit, pan and zoom the model; **Reset view** returns to the default camera.
- Zones are coloured by the chosen metric (green to red). **Boxes** show force boxes with depth following the value; **Labels** toggles zone labels; **Zone dims** shows zone widths.

- **Wind flow** shows streamlines. Each streamline takes the colour of the zone it passes and fades downstream.
- Size dimensions (X, Y, height) are boxes on dimension lines: click one and type a new value.
- Funnelling tags on each wall (9.5.7).
- A **north dial** turns with the camera. Its amber arrow marks where the wind comes from. The model itself is drawn in the frame of the wind case shown (wind always along +z), so the dial shows true north relative to that.
- **3D map** swaps the model for the Google photorealistic 3D map of the site with the same zones and streamlines laid over it.

9.6.4 Results

Eight numbered sheets for the chosen direction, each row giving symbol, formula, numbers substituted, result and clause reference:

1. Basic wind velocity at the site
2. Peak velocity pressure
3. Geometry and zone sizes
4. External pressure on every zone
5. Internal pressure
6. Worst net pressure for cladding and fixings
7. Structural factor
8. Overall force

Below them are an "All directions" summary and a "Where the numbers come from" table (source and use of each data set).

The right-hand drawer's **Site and wind climate** card repeats the hand-calc chain in order: $v_{b,map}$, A, z, c_{alt}, c_{season}, c_{prob}, v_b, c_{dir} per direction, $c_{e,T}$, $c_e(z)$, q_b, q_p, the governing q_p and the funnelling sentence. A small **?** on a line jumps to, opens and flashes the input that feeds it.

9.6.5 Compare

Normal w_e against funnelling $w_{e,fun}$, zone by zone, drawn on one scale.

9.6.6 Charts

Interactive redraws of the exposure factor $c_e(z)$, town correction $c_{e,T}$, $q_p(z)$, c_{dir}, wall and roof c_{pe}, c_{pi}, c_{alt}, c_{prob} and c_{season}. Hover to read; the amber points are this design. Each has a **?** help note.

9.7 Outputs

Quantity	Where	Units
$v_{b,map}$, v_b, q_b	Results 1, site card	m/s, kN/m²
$c_e(z)$, $c_{e,T}$, $q_p(z)$	Results 2, charts	-, kN/m²
Zone c_{pe}, w_e, `area`, `force` per wall and roof zone	Results 4, plan table, 3D	-, kN/m², m², kN
c_{pi}, w_i	Results 5	-, kN/m²
Worst net local pressure (zone, c_{pi}, direction)	Results 6	kN/m²
$c_s c_d$	Results 7	-
Overall horizontal force F_w per direction	Results 8	kN
Roof uplift per direction	Results, port data	kN
Governing wall pressure, wall suction and roof uplift	Simple report	kN/m²

9.8 Reports

The header **Report** button opens one wind report (the older separate detailed menu is retired in the header; the node card report still builds the combined report data).

Simple report (office hand-calculation layout, produced through the shared Civil ET module-report builder, so Word and text output follow the same format as the other nodes):

- Headline strip with $q_p(h)$ and the governing wall pressure, wall suction and roof uplift.
- Sections: Site data, Building data, Wind calculation (the chain of 9.6.4), Peak velocity pressure by direction, one section per direction (c_{pe} and w_e by zone), Notes and any warnings.
- Views and Design Diagrams placed **after** the working: per direction, the walls on the plan with w_e by zone (funnelling walls marked) and the roof zones on their own diagram. These are redrawn on canvas, so every zone has a traffic-light box and label.
- Option **Include full working (appendix)**: adds the worked calculation. The governing (largest q_p) direction is given in full and the others show only what changes with direction.
- Subscripts and superscripts (for example `q_{p}`, `v_{b,map}`) are converted to real sub/superscript in Word.
- References: BS EN 1991-1-4:2005+A1:2010 and its UK National Annex.

The detailed worked report is still used when a project-wide node-map report is built; it gives the worked sections for every direction plus governing pressures with zone and face.

9.9 Checking against a hand calculation

The recommended routine is to read each step of the node's chain (Results tab or the site card) alongside your own working. The points where a hand calculation and the node most often part company are:

- $c_{e,T}$ and $c_e(z)$: the node reads them at the shortest fetch within the direction sector, not along one line, and includes tidal estuaries.
- c_{dir} for a rotated building: the node takes the maximum over the $\pm 45^\circ$ sector for each face normal; a hand value read at one bearing can differ.
- Zone E and the D/E pair for long-face and end winds depend on which dimension is d, so check h/d for each direction.
- $v_{b,map}$ is read from the isopleth field; a rounded map value will differ slightly.

9.10 Connections to other nodes

The wind node publishes its **governing horizontal force** as an output dot (per design). The output is resolved onto the building's own axes and passed as magnitudes `WLx` and `WLy` (kN), the wind load pair used by the receiving nodes. The receiver applies its own sign convention. A component under 1 N is zeroed as rounding noise. The wire also carries the wind result summary (design name, $v_{b,map}$, q_p at the top, $c_s c_d$, governing direction, F_w, roof uplift, worst net pressure and q_p/F_w by direction).

Allowed wires from the wind node:

To	Effect
Beam, Slab, Column, Foundation, Base plate	Wire labelled "WIND ACTIONS" feeds the wind load set (`WLx`, `WLy`).
Maths (math) node	As a value source.

The wind node has no input socket. A wind load from this node is a **characteristic action**; apply load combinations and factors in the receiving node.

9.11 Tips and limitations

- The node is for UK sites and the UK National Annex only.

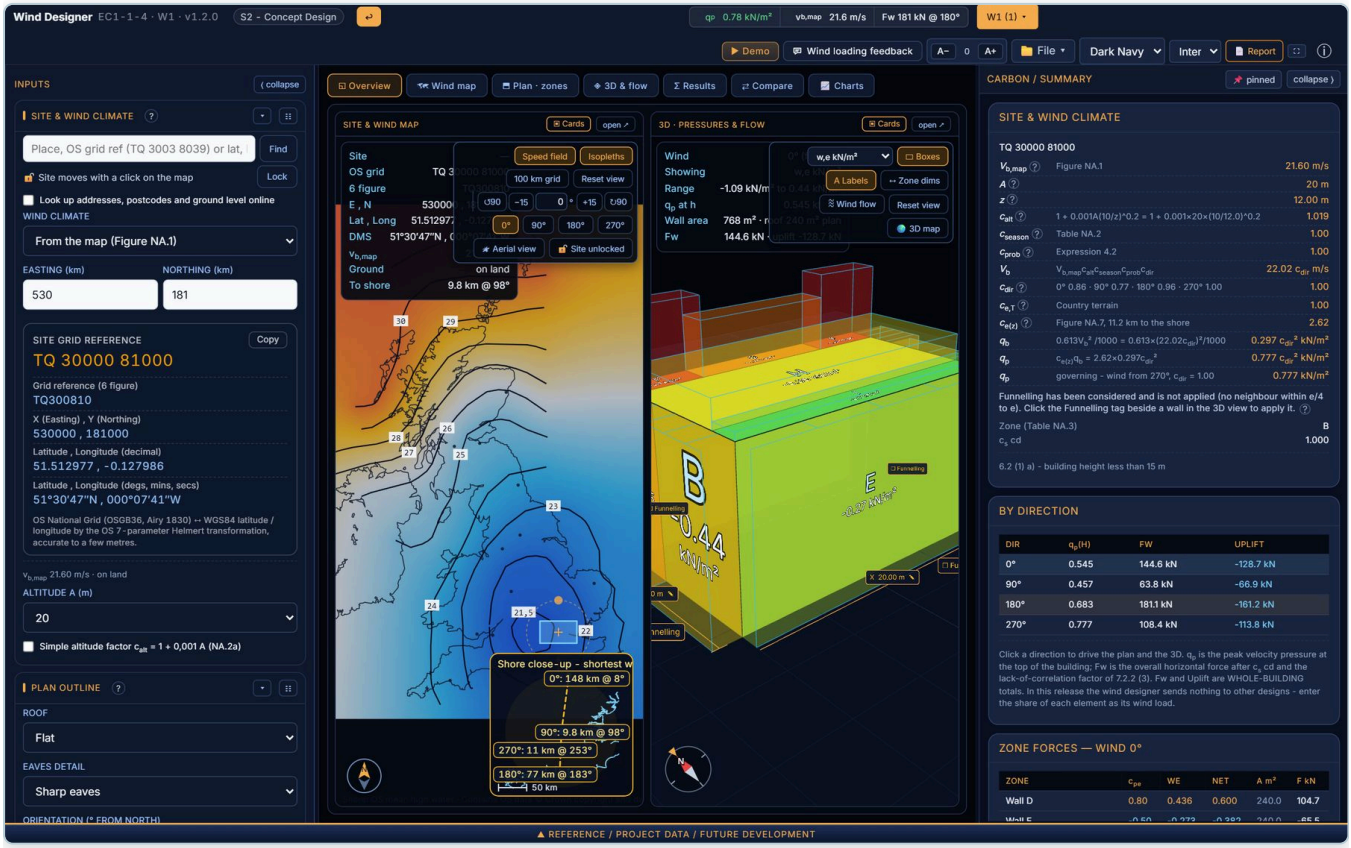
- Non-rectangular plans are an approximation by enclosing rectangle and need engineering review.
- c_d must be read from NA Figure NA.9 yourself (not reproduced); use Annex B if you want the node to derive it.
- For the overall force on a closed building you may switch internal pressure off; for cladding, fixings and uplift keep it on.
- The default "unknown openings" basis uses +0.2 and -0.3 and is the conservative choice.
- If you rely on an automatic fetch, check the shore rays on the map once; use the manual cap for Northern Ireland or any water the OS data misses.
- Keep the site locked once placed, then rotate the building to match the plot.
- Orography is only an input here (c_o); the node does not derive it from terrain.
- A fetch can change when the shore setting changes, which changes $c_e(z)$ and q_p. Re-read q_p before relying on earlier numbers.

> **References.** The standards, guidance and data sources used in this chapter are listed, with web links, in Chapter 17.

Screen reference - panel by panel

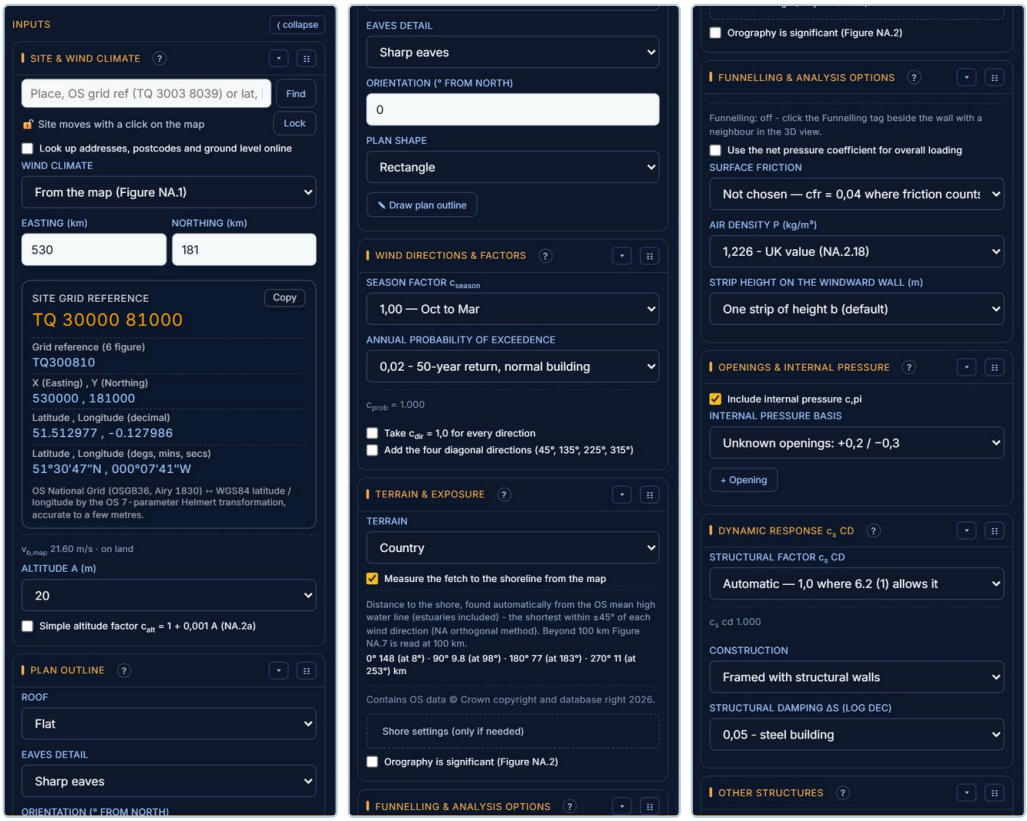
These screenshots show every panel of the module exactly as it appears on screen, scrolled from top to bottom, so you can match any field, button or result to the explanations in this chapter. Cards that are collapsed by default are shown expanded.

Inputs: the whole screen



Inputs: the whole screen.

Inputs: input panel (left of the screen)

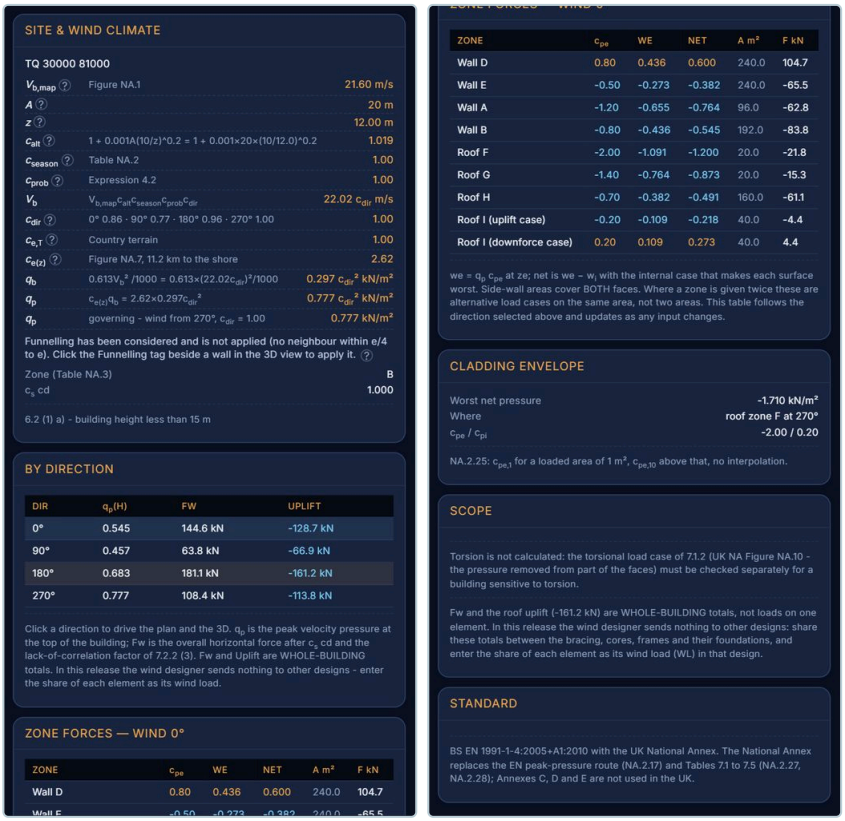


Inputs, input panel (left of the screen), parts 1-3 of 4. Cards shown: Site & wind climate, Plan outline, Wind directions & factors, Terrain & exposure, Funnelling & analysis options, Openings & internal pressure, Other structures.



Inputs, input panel (left of the screen), part 4 of 4. Cards shown: Openings & internal pressure, Other structures.

Inputs: results panel (right of the screen)



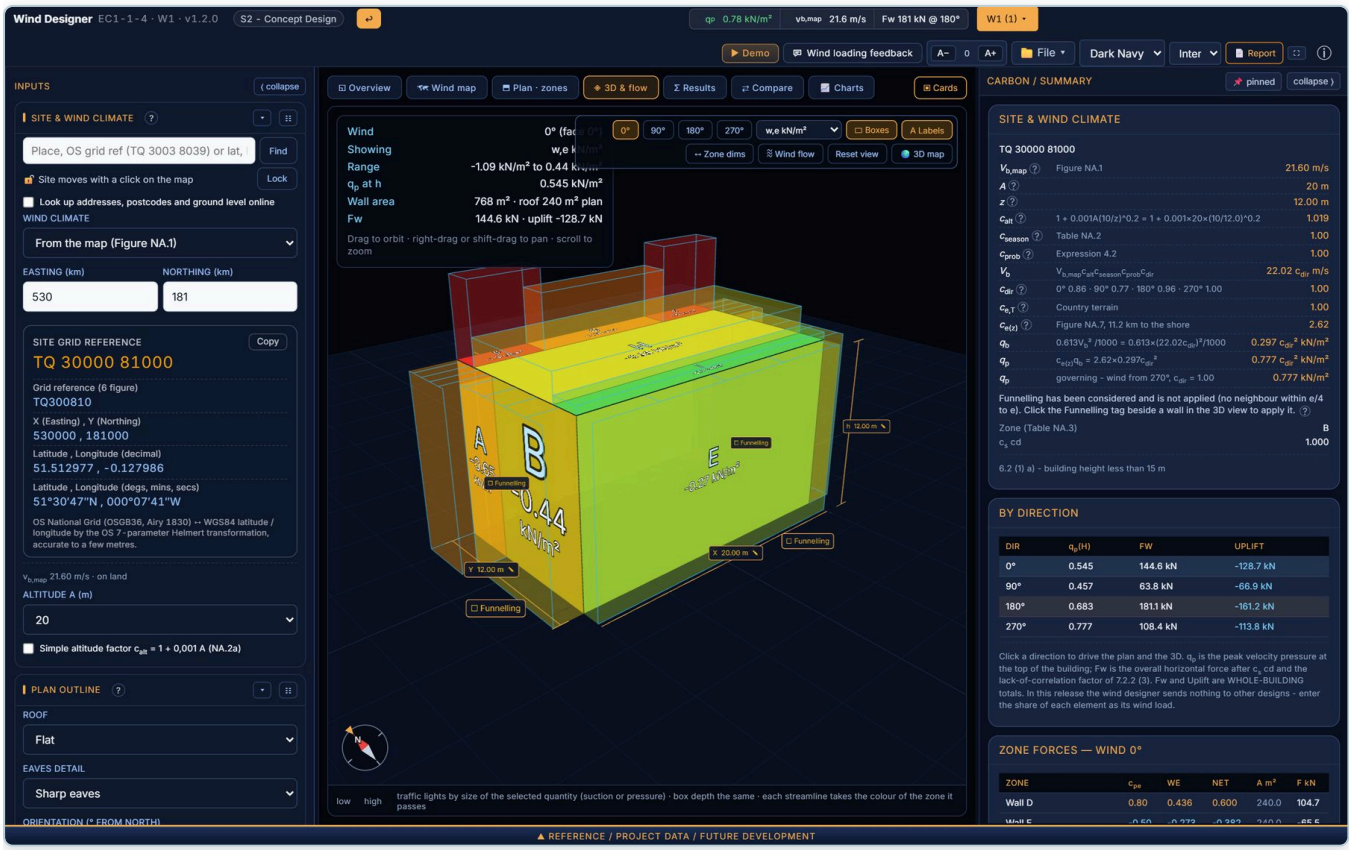
Inputs, results panel (right of the screen), parts 1-2 of 2. Cards shown: Site & wind climate, By direction, Zone forces — wind 0°, Cladding envelope, Scope, Standard.

Plan and zones tab: the whole screen



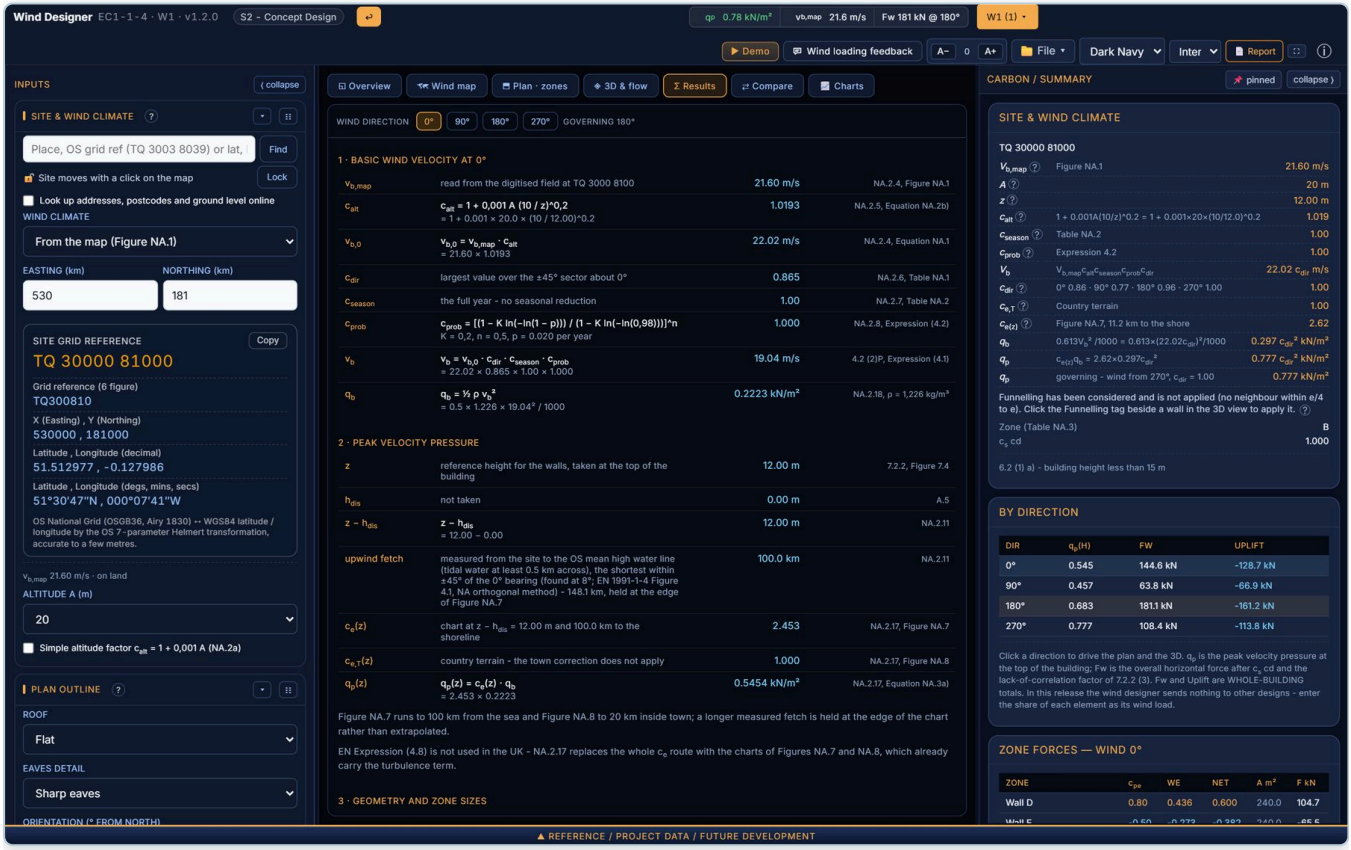
Plan and zones tab: the whole screen.

3D and flow tab: the whole screen



3D and flow tab: the whole screen.

Results tab: the whole screen



Results tab: the whole screen.

Results tab: working panel (centre of the screen)



WIND DIRECTION 0° 90° 180° 270° GOVERNING 180°

1 · BASIC WIND VELOCITY AT 0°

$v_{b,map}$	read from the digitised field at TQ 3000 8100	21.60 m/s	NA.2.4, Figure NA.1
c_{alt}	$c_{alt} = 1 + 0{,}001\ A\ (10 / z)^{0,2}$ = 1 + 0.001 × 20.0 × (10 / 12.00)^0.2	1.0193	NA.2.5, Equation NA.2b)
$v_{b,0}$	$v_{b,0} = v_{b,map} \cdot c_{alt}$ = 21.60 × 1.0193	22.02 m/s	NA.2.4, Equation NA.1
c_{dir}	largest value over the ±45° sector about 0°	0.865	NA.2.6, Table NA.1
c_{season}	the full year - no seasonal reduction	1.00	NA.2.7, Table NA.2
c_{prob}	$c_{prob} = [(1 - K \ln(-\ln(1 - p))) / (1 - K \ln(-\ln(0{,}98)))]^n$ K = 0,2, n = 0,5, p = 0.020 per year	1.000	NA.2.8, Expression (4.2)
v_b	$v_b = v_{b,0} \cdot c_{dir} \cdot c_{season} \cdot c_{prob}$ = 22.02 × 0.865 × 1.00 × 1.000	19.04 m/s	4.2 (2)P, Expression (4.1)
q_b	$q_b = ½\ \rho\ v_b^2$ = 0.5 × 1.226 × 19.04² / 1000	0.2223 kN/m²	NA.2.18, ρ = 1,226 kg/m³

2 · PEAK VELOCITY PRESSURE

z	reference height for the walls, taken at the top of the building	12.00 m	7.2.2, Figure 7.4
h_{dis}	not taken	0.00 m	A.5
$z - h_{dis}$	$z - h_{dis}$ = 12.00 − 0.00	12.00 m	NA.2.11
upwind fetch	measured from the site to the OS mean high water line (tidal water at least 0.5 km across), the shortest within ±45° of the 0° bearing (found at 8°; EN 1991-1-4 Figure 4.1, NA orthogonal method) - 148.1 km, held at the edge of Figure NA.7	100.0 km	NA.2.11
$c_e(z)$	chart at $z - h_{dis}$ = 12.00 m and 100.0 km to the shoreline	2.453	NA.2.17, Figure NA.7
$c_{e,T}(z)$	country terrain - the town correction does not apply	1.000	NA.2.17, Figure NA.8
$q_p(z)$	$q_p(z) = c_e(z) \cdot q_b$ = 2.453 × 0.2223	0.5454 kN/m²	NA.2.17, Equation NA.3a)

Figure NA.7 runs to 100 km from the sea and Figure NA.8 to 20 km inside town; a longer measured fetch is held at the edge of the chart rather than extrapolated.

EN Expression (4.8) is not used in the UK - NA.2.17 replaces the whole c_e route with the charts of Figures NA.7 and NA.8, which already carry the turbulence term.

3 · GEOMETRY AND ZONE SIZES

Results tab, working panel (centre of the screen), part 1 of 5. Cards shown: 2 peak velocity pressure, 3 geometry and zone sizes.



WIND DIRECTION 0° 90° 180° 270° GOVERNING 180°

3 · GEOMETRY AND ZONE SIZES

b	breadth of the face across the wind	20.00 m	7.2.2
d	depth of the building along the wind	12.00 m	7.2.2
h	height to roof level	12.00 m	7.2.2
h/d	h / d = 12.00 / 12.00	1.000	Table NA.4 is entered with h/d
e	e = min(b, 2h) = min(20.00, 2 × 12.00)	20.00 m	7.2.2 (2), Figure 7.5
wall A	on each side face, from 0.00 m to 4.00 m behind the windward edge	4.00 m × 12.00 m high	Figure 7.5
wall B	on each side face, from 4.00 m to 12.00 m behind the windward edge	8.00 m × 12.00 m high	Figure 7.5
roof F	across the wind 0.00 to 5.00 m, along the wind 0.00 to 2.00 m	10.0 m²	Figure 7.6
roof F	across the wind 15.00 to 20.00 m, along the wind 0.00 to 2.00 m	10.0 m²	Figure 7.6
roof G	across the wind 5.00 to 15.00 m, along the wind 0.00 to 2.00 m	20.0 m²	Figure 7.6
roof H	across the wind 0.00 to 20.00 m, along the wind 2.00 to 10.00 m	160.0 m²	Figure 7.6
roof I	across the wind 0.00 to 20.00 m, along the wind 10.00 to 12.00 m	40.0 m²	Figure 7.6

The zones are set out once, from e, and the same rectangles drive the plan drawing, the 3D model and the forces below, so the three cannot disagree.

4 · EXTERNAL PRESSURE ON EVERY ZONE

we	$we = q_p(ze) \cdot c_{pe}$ worked zone by zone below	see table	5.2 (1), Expression (5.1)
$F_{w,e}$	$F_{w,e} = we \cdot Aref$ Aref is the zone area from step 3	see table	5.3 (2), Expression (5.5)

SURFACE	ZONE	c_{pe}	q_p (kN/m²)	WE = q_p × c_{pe} (kN/m²)	A (m²)	F = WE × A (kN)	NOTE
Windward	D	0.80	0.5454	0.5454 × 0.80 = 0.4363	240.0	0.4363 × 240.0 = 104.7	ze = 12.00 m
Leeward	E	-0.50	0.5454	0.5454 × -0.50 = -0.2727	240.0	-0.2727 × 240.0 = -65.5	ze = h
Side walls	A	-1.20	0.5454	0.5454 × -1.20 = -0.6545	96.0	-0.6545 × 96.0 = -62.8	both faces together
Side walls	B	-0.80	0.5454	0.5454 × -0.80 = -0.4363	192.0	-0.4363 × 192.0 = -83.8	both faces together
Roof	F	-2.00	0.5454	0.5454 × -2.00 = -1.0909	20.0	-1.0909 × 20.0 = -21.8	uplift

Results tab, working panel (centre of the screen), part 2 of 5. Cards shown: 3 geometry and zone sizes, 4 external pressure on every zone.



Results tab, working panel (centre of the screen), part 3 of 5. Cards shown: 5 internal pressure, 6 worst net pressure for cladding and fixings, 7 structural factor.

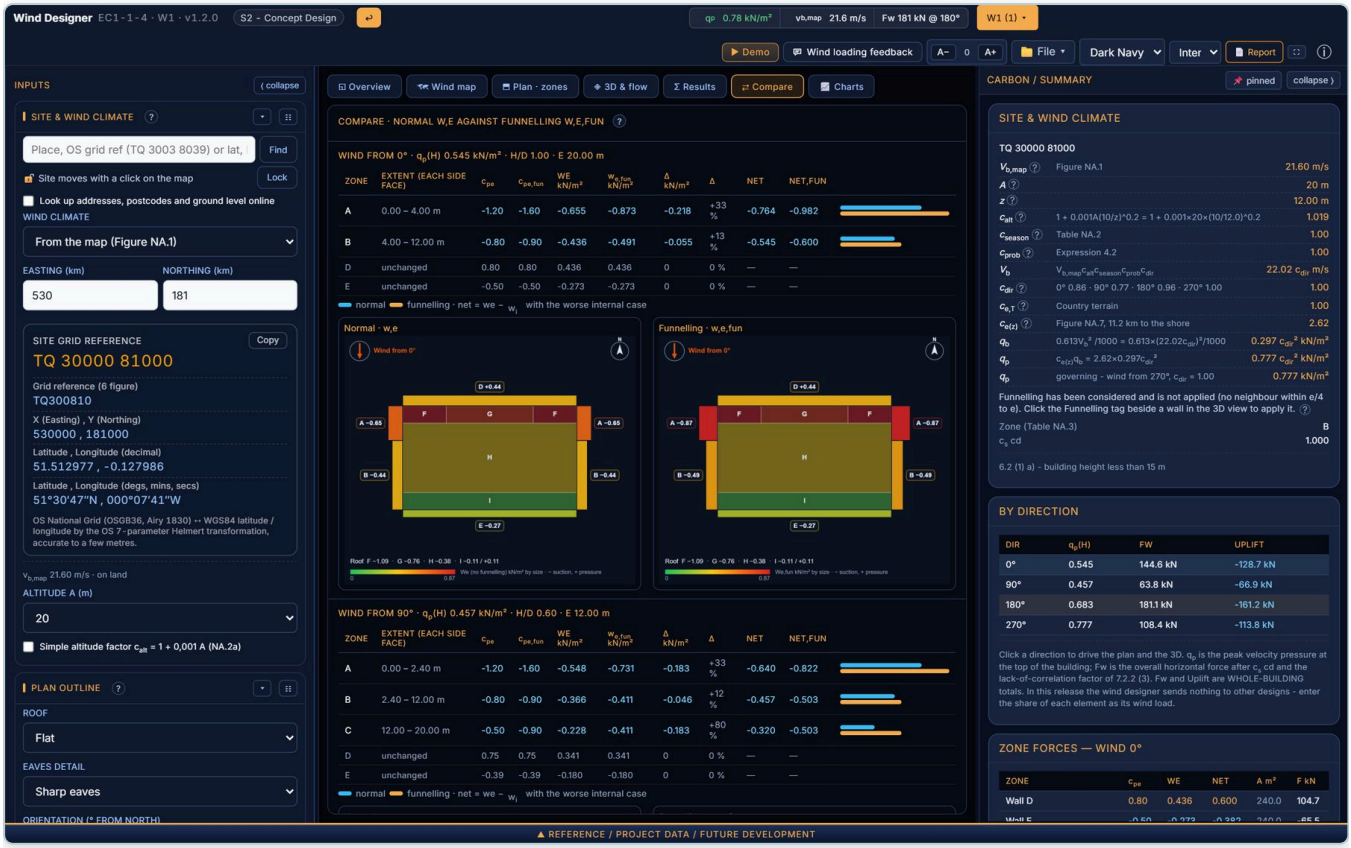


Results tab, working panel (centre of the screen), part 4 of 5. Cards shown: All directions, Where the numbers come from.

Figure 7.13 c_{pi}	Both curves digitised from BS EN 1991-1-4: c_{pi} = 0,35 at mu ≤ 0,33, falling to -0,3 at mu = 0,90 for h/d ≤ 0,25 and to -0,5 at mu = 0,95 for h/d ≥ 1,0.
Figure 7.23 $c_{f,0}$	Rectangular sections: 2,0 to d/b = 0,2, peak 2,4 at d/b = 0,7, then 2,1 / 1,65 / 1,0 / 0,9 at d/b = 1 / 2 / 5 / 10 and above.
Figure 7.36 psi-lambda	The five solidity curves digitised from the figure. The figure itself calls these values indicative.
Tables NA.1 to NA.10, 7.9, 7.10	Transcribed from the printed tables.

Results tab, working panel (centre of the screen), part 5 of 5. Cards shown: All directions, Where the numbers come from.

Compare tab: the whole screen



Compare tab: the whole screen.

Compare tab: working panel (centre of the screen)



Compare tab, working panel (centre of the screen), part 1 of 3. Cards shown: Compare normal w,e against funnelling w,e,fun.

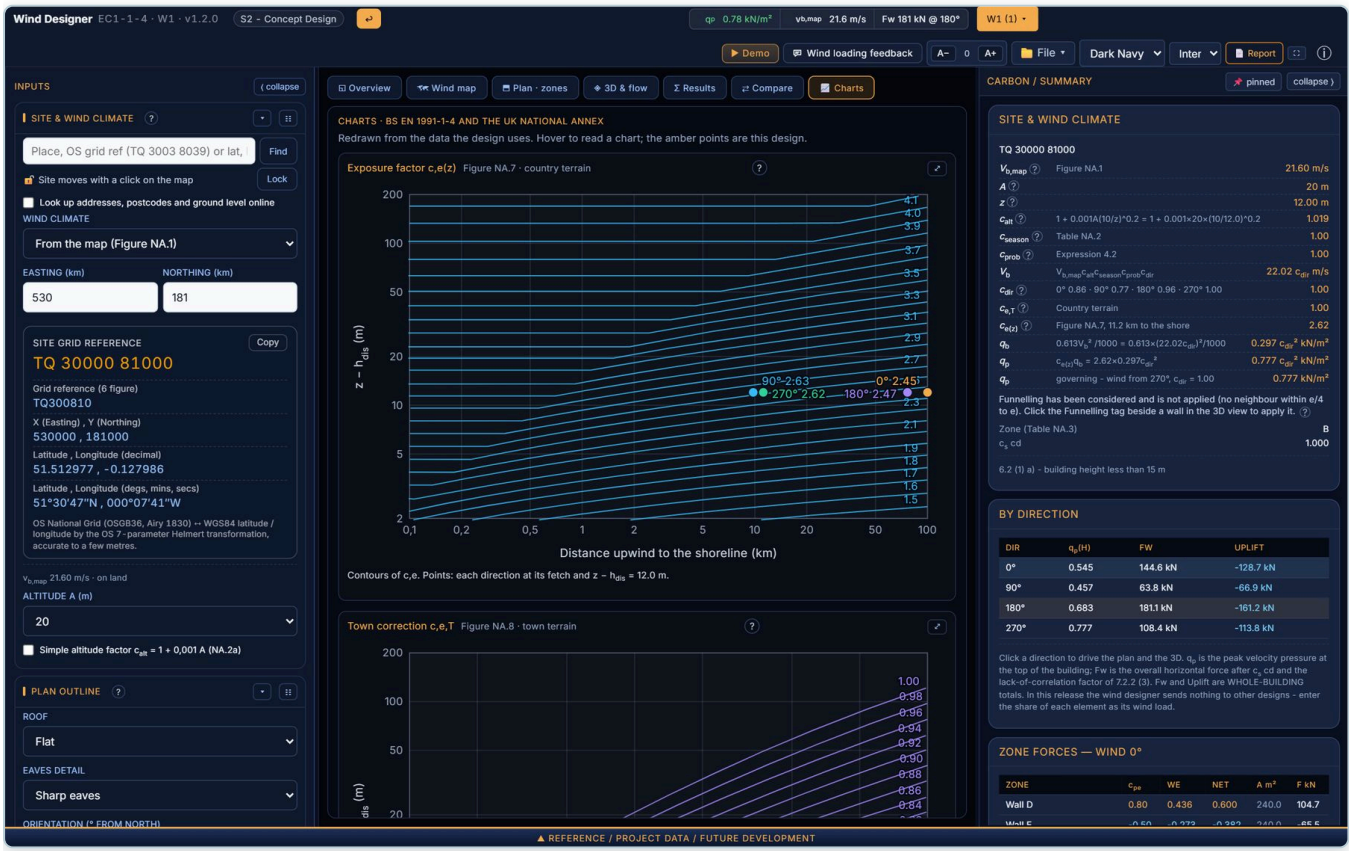


Compare tab, working panel (centre of the screen), part 2 of 3.



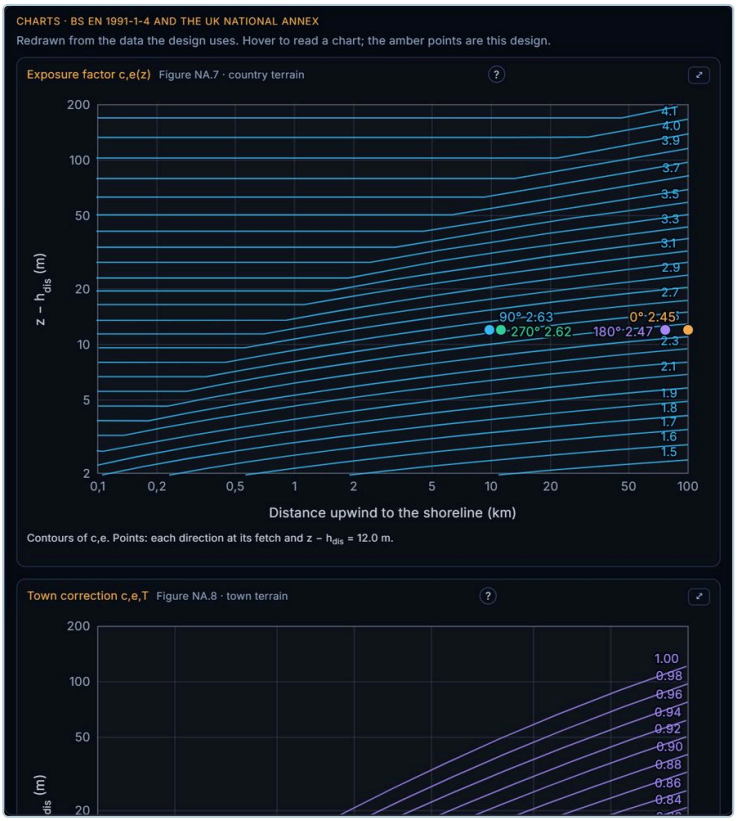
Compare tab, working panel (centre of the screen), part 3 of 3.

Charts tab: the whole screen

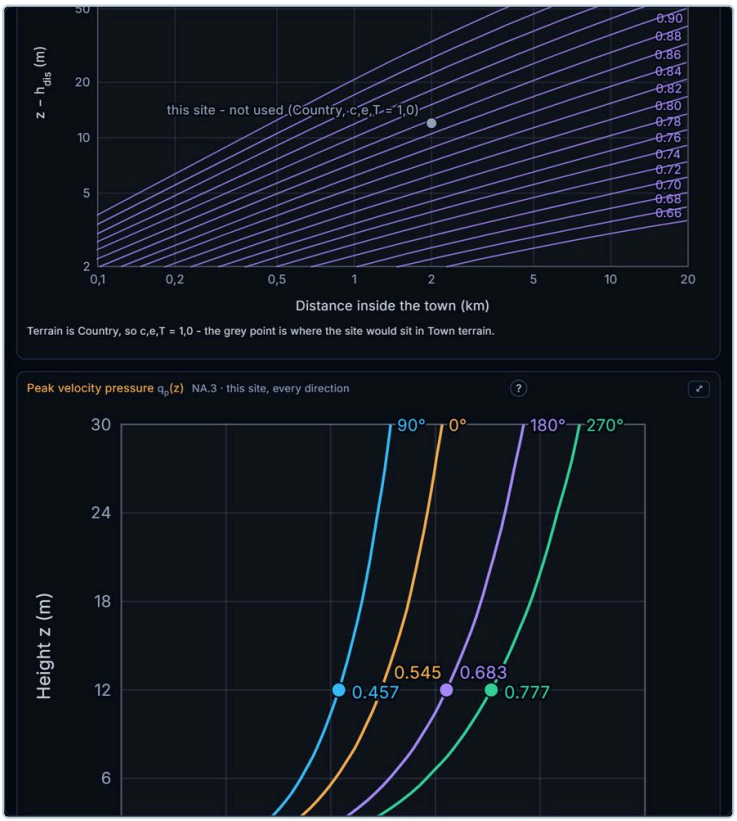


Charts tab: the whole screen.

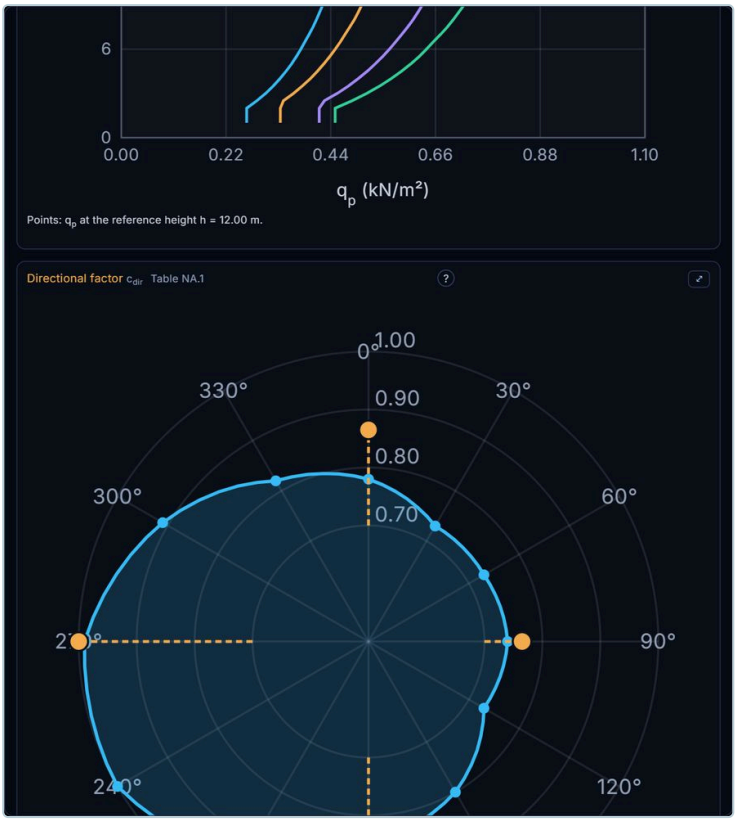
Charts tab: working panel (centre of the screen)



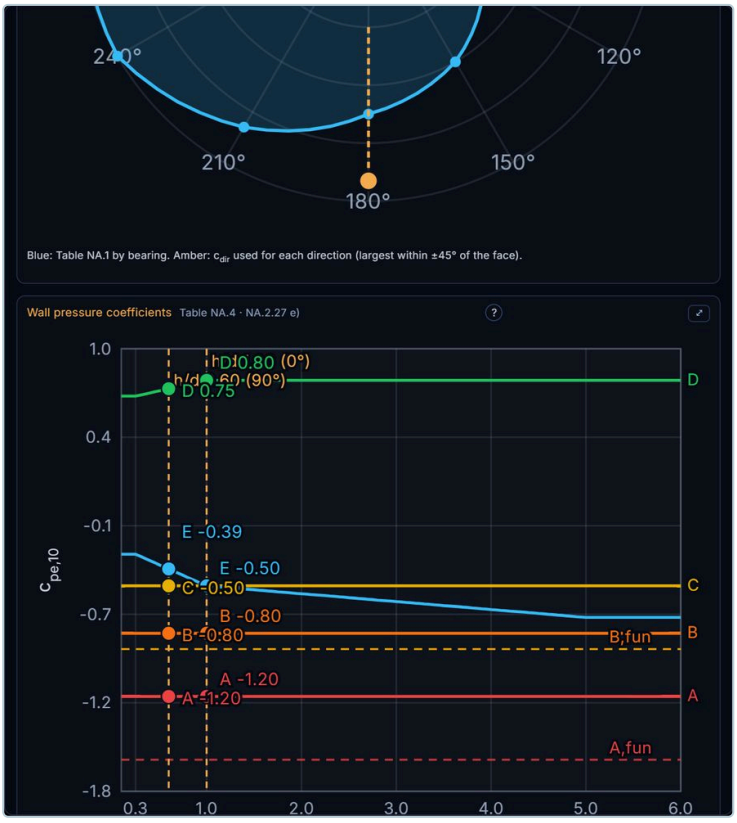
Charts tab, working panel (centre of the screen), part 1 of 4.



Charts tab, working panel (centre of the screen), part 2 of 4.



Charts tab, working panel (centre of the screen), part 3 of 4.



Charts tab, working panel (centre of the screen), part 4 of 4.

Wind map tab: the whole screen



Wind map tab: the whole screen.

CHAPTER 10

Sketch-ET

Sketch-ET is the concept-stage scheme designer. You trace a column grid, beams, slab panels, footings and portal frames over a PDF, DXF or image of the architect's plan, say what the floors are made of and what they carry, and it returns indicative member sizes, column reactions, quantities and an embodied-carbon figure. A 3D view sits beside the plan. The scheme can be exported as an IFC4 model, handed to the IFC node for a carbon study, or sent on as column reactions to the Column, Foundation, Base plate, Beam and Slab nodes.

The node card is titled **Sketch-ET** and is tagged "Sketch - PDF/DXF".

Read this first. Sketch-ET produces a *scheme*, not a design. Every report it issues carries this statement: sizes come from published span/depth thumb rules and a per-beam continuous-beam analysis of tributary loads, not from a 3D frame analysis; there is no lateral stability design, no wind or notional horizontal force, no imposed-load reduction (EN 1991-1-1 6.3.1.2), no second-order effects, no deflection or vibration serviceability check (the portal frame design is the exception, see 10.6.5), and nothing on fire, connections, robustness or foundations beyond a bearing-pressure sizing. Every size must be confirmed by full design before it leaves the office.

10.1 What it does

1. Takes a plan underlay (PDF, DXF or image), which you calibrate to scale.
2. Lets you draw a setting-out grid, columns, beams, slab panels (rectangles or polygons), footings under columns, and portal frames; you can also drop in ready-made building and portal-frame templates.
3. Works out the slab load (system self-weight plus superimposed dead load plus an imposed-load category), sheds it onto the beams, and solves every beam as a real continuous beam.
4. Sizes the slab depth, every beam, every column and every footing, aiming at a target utilisation, and bands each member Green / Amber / Red.
5. Designs portal frames (frame analysis, sections, haunches, purlins and bases) with a quick live design and a Final design check.
6. Produces a quantity take-off and an embodied-carbon figure (A1 to A5) with a SCORS band.
7. Reports to Word or PDF, exports IFC4, and sends column reactions down the wires.

Scope and level of detail are limited on purpose: "one floor plate repeated" is the basic idea, with storeys and per-column overrides, and no lateral system is designed (bracing and shear walls are placed and positionally checked, not designed).

10.2 Opening it

- On the node map add a **Sketch-ET** node (Concept scheme). The card lists the node's *designs* (P1, P2 and so on). Each row is one 3D design. Select a row, rename it, copy it with the copy button on the row, or use **+ Add Sketch design** to add another starting from defaults.
- The card has **Run** (`Run P1` / `Re-run P1`) for the selected design. A design that has not been run is not calculated, and sends no results down its wires (its wire says "not run yet"). **Open Sketch-ET** opens the module and also runs the design. Double-clicking the card opens it too.
- The card shows materials (steel tonnes and concrete m^3) and A1 to A5 CO_2e for the designs that have been run. Designs can be excluded from the project carbon total.
- Inside the module, a design picker in the header switches between designs.

10.3 The screen

- **Ribbon** across the top with nine tabs (10.4). The ribbon folds away (button at its right) to give the drawing the height; folded, a tab opens its tools as a drop-down. Below about 1360 px of stage width the ribbon switches to

"tight mode" (icons only). A short window starts folded.

- **Stage** views, chosen on the View controls: **Plan**, **Split** (plan and 3D side by side), **3D**, **Elevation**, **Portal compare** (only when the design has a portal), **Results**, and **Working**. The split ratio is draggable.
- **Properties** dock: a card that edits whatever is selected (sections, grades, spans, ends, loads, "Apply to every..." for copying a decision to the other members).
- **Right drawer**: Quantities and Summary Totals tiles, life-cycle modules A1 to A5, the SCORS band, element carbon split, and quantity and embodied-carbon donuts.
- **Templates** column on the left of the view (collapsible): building and portal-frame templates (10.7).
- A **status line** reports what the last action did and what the tool expects next.
- A readout (**HUD**) over the view shows SCORS, loads and the governing member; it can be switched off.

Per-session habits (ribbon tab, split, panel states) are stored in your browser. The design itself is stored in the project.

While you are typing in any box, recomputes and repaints are held until you leave the box (Tab, Enter or click away), so numbers do not move under the caret.

10.4 Ribbon tabs

Tab	Contents
Underlay and grid	Sheet menu (page, scale, lock, position), grid line and level line tools, the Underlay card, the Setting-out grid card
Building	Design name, lateral stability choice, GIA override, floor-area/levels/GIA read-outs, and the **Site location (UK)** card
Materials and factors	γ_G (default 1.35) and γ_Q (default 1.50). Grades are set on the member, not here.
Draw	Tools: Select, Column, Beam, Slab, Slab (poly), Footing, Portal, Erase, Pan; Comment and Text box; Edit group (undo, redo, delete, fit); **Levels** bar
Edit	Move by (dX, dY, dZ), Copy by (count), Move to (X, Y, Z), Nudge arrows, Rotate, Delete, Undo/Redo
Dimension	Linear, Aligned, Radius, Angle dimensions; Area, Calibrate, Measure 3D; sheet scale chip
View	Elevation tool and where it sits; Snaps menu; Show chips; Lock (Grid, Frame); Forces (ULS or SLS, diagrams); Colour mode, projection and 3D view
Tune	Display sliders (model opacity, slab opacity, annotation size, underlay opacity, 3D grid), Sizing (target utilisation, long-span flag), thumb-rule bands
Export and IFC	Report options, Word and PDF report, IFC export, send to IFC node, open IFC node, pull carbon back

10.5 Setting up

10.5.1 Underlay

On the Underlay card choose **Load a PDF, DXF or image**.

- PDF pages are drawn in the browser. If the preview does not appear, export the plan as PNG or DXF instead.
- A multi-page PDF lets you choose the page. Each level can have its own sheet: load one while a level is selected and it belongs to that level; use **Put this sheet behind every level** to share one.
- A DXF is read as line segments, already in metres via a DXF-units-per-metre factor. Lines from a DXF, and the vectors traced out of a PDF, are snappable (the "Drawing lines" snap).
- **Calibrate**: click two points on the drawing and type the real distance. Check a second known dimension before trusting the scale. A sheet that replaces another keeps the previous scale and position, and the card says so.
- **Remove** and **Replace** ask for confirmation; a removed sheet is kept for the session so it can be put back.

- The sheet has opacity (default 0.55) and a lock (default locked). The underlay image itself is stored in the design, so keep sheets small.

10.5.2 Setting-out grid

The grid is a set of drawn grid lines (any position, angle and length). You can draw them with the Grid line tool or from the Setting-out grid card, which keeps the grid as bay *spacings* (defaults: three X bays of 6 m, two Y bays of 6 m; optional $45°$ diagonals; labels and bay dimensions on or off). Editing a bay moves every line beyond it. Grid lines select, delete and undo like any other element and can be locked (Lock > Grid on the View tab).

The **snap grid** (default 0.25 m) is separate from the setting-out grid.

10.5.3 Building

Input	Default	Range / notes
Storeys (suspended floors carried by a column)	1	1 to 120. Set on the Levels bar of the Draw tab.
Storey height	3.5 m	1.8 to 20 m. Per level on the Levels bar.
Roof carried as one extra level	off	Roof toggle on the Levels bar.
Concrete f_{ck} (fallback)	30 N/mm²	20 to 90. Real grades are set per member.
Steel f_y (fallback)	355 N/mm²	235 to 690.
Target utilisation	0.85	0.55 to 1.00. Per-element overrides exist (beam, column, slab, footing, portal) on the Tune tab.
γ_G, γ_Q	1.35, 1.50	EN 1990 Table NA.A1.2(B), expression 6.10
Long-span flag	14 m	A beam longer than this is flagged, not refused.
GIA override	blank	Blank uses drawn slab area times levels. SCORS divides by this.
Lateral stability	Not set yet	Braced frame / RC shear walls or core (not modelled) / Moment frame (not checked).

Levels: the **+** and **-** on the Levels bar add and remove storeys (the count lives in one place). A level can be named, given its own height and its own sheet. A copy of a floor can be pasted to the level above, and pasting above the top level grows the storey count.

10.5.4 Slab system, loads and defaults

Slab and load choices live in the design and the Properties card:

Input	Default
Slab system	RC solid slab on beams
Thickness	250 mm
Continuous	yes
Span direction	auto (or X or Y)
Superimposed dead load	picks from the dead-load library: 50 mm screed, ceiling and services, light partitions
Imposed-load category	B1 (office) plus an extra kN/m² if wanted
Movable partitions	0 kN/m² (the dead-load library already offers a partition allowance; entering both double-counts)

Slab systems on offer: RC solid slab on beams; RC flat slab (no beams); composite metal deck; ComFlor 80 deep composite deck; precast hollowcore with topping; timber joisted floor; plus one row per Tata Steel ComFlor and RoofDek profile (transcribed from the manufacturer's manuals, with page references; ComFlor 46, 51+, 60, 60 closed ends, 80, 80 closed ends, 100, 210 and 225 and a range of RoofDek profiles). Hollowcore units are tabulated at 150 to 450 mm with a 75 mm topping assumed. The deck libraries are for concept-scheme drawing against real profiles; they are not a composite deck design and carry no fire case, vibration check or propping schedule. Use the ComFlor slab node for that.

New elements take their material and section defaults (steel UB, 355 grade; square column 300 by 300 mm; and so on) from the model defaults, which are written by **Apply to every...** in the Properties card.

Beam materials: steel, reinforced concrete, glulam timber. Column shapes: rectangular, square or circular RC, or a steel section (UC). Beam ends: Pinned (default for steel and timber), Continuous (default for RC), Roller, Fixed.

10.6 Drawing and working

10.6.1 Tools

Tool	Notes
Select	Click, marquee, Ctrl-click to add. Selection is scoped to the floor shown.
Column	Click to place. New columns line up with existing ones when Square snap is on.
Beam	Click start and end; type a length and press Enter during the draw.
Slab / Slab (poly)	Slab drags a rectangle; Slab (poly) clicks the corners and closes on the first point or Enter.
Footing	Toggles a footing under the clicked column. A footing belongs to one column; if the column is deleted the footing is dropped.
Portal	Draw the line of the frame span; eaves height, rise, spacing and bays are properties of the frame.
Erase	Deletes what you click.
Pan	Or hold Space.
Comment / Text box	Mark-ups with or without a leader. They carry no engineering meaning.
Dimension tools	Linear, aligned, radius, angle; dimensions are elements with ids and can be moved clear of the drawing.
Area	Measures an area.
Calibrate	Sets the sheet scale from a known dimension.
Elevation	Draws a view line and box on the plan. Double-click it to open the elevation.
Grid line, Level line	Level lines exist only in an elevation.

10.6.2 Snapping

The Snaps menu on the View tab has a master switch and individual snaps: Grid; Ends and centres; Midpoints; Drawing lines (the imported sheet); **Square - 90 degrees only** (default on); Angle 0/45/90 (used only when Square is off); Dimensions in line. A real object (a column centre, a beam end, a sheet line) beats the Square lock; plain grid rounding does not.

10.6.3 Elevations and the work plane

An elevation is a view *along a line* drawn on the plan, for what a plan cannot show: a pitched roof, a raised or sloping beam, a portal frame. It has a grid of its own and uses the ribbon's tools (column, beam, footing, portal, erase, pan, grid line, level line); slab, comment and dimension tools are disabled there. Members drawn in an elevation land on the

elevation's **work plane**, not on the view line, so an elevation drawn outside the building still joins the columns it was snapped to; objects off the plane draw faint. A finite/infinite toggle clamps or frees pan and zoom. The View tab can put the elevation in the split and attach its own PDF or image sheet behind it.

10.6.4 Editing and selection

Edit tab: Move by (dX, dY, dZ in metres, to the level nearest the height), Copy by (count), Move to (absolute X, Y, Z of the reference point), Nudge arrows, Rotate, Delete, Undo, Redo. A member can be moved either by typing the numbers or by picking from-and-to points on screen.

Keyboard (not active while typing in a box):

Keys	Action
Ctrl+Z / Ctrl+Y	Undo / redo, per change, in plan and 3D
Ctrl+A	Select everything on the floor shown (every floor if "every level" is on)
Ctrl+C, Ctrl+V, Ctrl+D	Copy, paste, duplicate
Delete / Backspace	Delete the selection
Arrow keys	Nudge the selection by the snap grid; Shift for 10 times
F	Fit the view
Space (held)	Temporary pan
Esc	First cancels a part-drawn item, then drops the tool back to Select, then clears the selection
Enter	Closes a polygon slab or a dimension chain
H H	Hide the selection from the 3D
U H	Unhide everything
M M	Move: point a start and end on screen
C C	Copy: place copies from a point
R R	Rotate about a picked point (S toggles snap while rotating)

For H H, U H, M M, C C and R R type the two letters within a second. During a carried move or copy, X or Y restricts the direction and digits give a distance; Enter confirms and Esc cancels. Right-click menus on the plan and 3D offer: copy/paste at the level being drawn on, **Select this whole floor**, **Select every...**, **Hide the selection**, **Show the hidden items**, **Section box**, **Brace this bay**, **Shear wall in this bay**, **Copy this floor to the level above**, **Open elevation**, snapping, and many portal-frame options (apex haunch, ties, hit-and-miss, join into one multi-span frame, split into single-span frames).

When several members are selected, a change of type applies to every selected member. The frame can be locked (Lock > Frame) while marking up, and double-clicking an empty part of the 3D recentres the model origin.

10.6.5 Portal frames

A portal frame is a whole frame drawn as its span line, not a beam, so it is analysed separately. Its card holds: eaves height, apex rise (with a chart-limits check), spans (multi-span frames are supported), bay count and bay widths or frame spacing, pitch, bases (pinned or fixed), haunch length and depth, optional apex haunch, hit-and-miss frames with valley beams, purlin and side-rail restraint, deck profile, loads and deflection limits.

Loads on a frame (defaults): g_k 0.35 kN/m² (templates use 0.25 plus the deck and purlins), q_k 0.60 kN/m², and typed net wind **uplift** and **horizontal** pressures. These wind pressures are typed, not derived. Use the Wind node (Chapter 9) for the actual EN 1991-1-4 analysis, then type or compare the values. Loads may be per m² of slope or of plan.

Design routes: a **quick** live design (elastic frame analysis, member, rafter-zone, haunch, deflection and miss-frame checks, with a plain upward section search) updates as you edit; the **Final design check** button runs the full search (including a plastic route and lighter-neighbour trim) once and stores the chosen sections with the inputs they were chosen for. Change anything that affects the frame and it returns to the quick design, marked as such. The **Portal compare** view shows a frame designed both ways side by side. The plastic method follows the closed-form portal statics behind the Weller-style charts (H/WL, M_p/WL^2 ratios, with a stated validity range for span/height 2 to 10 and pitch 6° to 20°), and the lateral-torsional checks refer to SCI P399 (restraint and C_1 values). Footings under portal legs also check eccentric bearing, uplift against 0.9 times the pad weight, and sliding (friction coefficient from a 20° interface angle, or the base thrust passed to a tie or ground slab).

10.6.6 Footings

Four types: pad, strip, raft (local panel), pile cap. Defaults: allowable bearing 150 kN/m² and thickness 0.6 m. Spread footing area comes from P/A with the footing's own weight; the check is on characteristic load. Pile groups are 1, 2, 3, 4, 5, 6, 7 or 9 piles. Footings are marked PF on the plan. No sliding (except portal bases), overturning, punching, bending, reinforcement or settlement is checked for ordinary footings, and the report says so.

10.7 Templates

The Templates column offers:

- **Buildings** - Office (steel, composite deck), Office (RC flat slab) and Residential (RC flat slab), each in three sizes. A template is ordinary model: drawn grid lines, a column at every intersection, beams, slabs and a pad under each column. Its bays are editable in input boxes on the plan and 3D; changing one stretches the building (lines beyond it move, and members in the bay scale with it). **+** and **-** add or remove a bay at the far end, and floors can be added or deleted. Floor systems on offer: RC flat slab, RC slab on beams, composite deck (ComFlor 60) on steel beams, precast hollowcore on beams.
- **Portal frames** - Single span, Two-span and Hit-and-miss three-span sheds at a range of spans, each with five 6 m bays at 6° pitch and pinned bases. A template goes in with its final plastic design already stored, provided the model uses the target utilisation and partial factors the catalogue was designed for (0.85, 1.35, 1.50); otherwise the live design is used. Template loads are a placeholder, not a wind and snow assessment.

10.8 Calculation basis

The sizing basis is cited against published sources rather than a design code:

Element	Basis
Slab depth	Span/depth rule of the chosen system; or a thumb-rule band if you take the Tune sliders.
Thumb rules	Cobb, *Structural Engineer's Pocket Book* (4th edition): RC slabs and beams p.121, steel p.213, composite p.247, timber p.263. Examples: steel secondary beam $L/15$ to $L/25$, primary $L/10$ to $L/15$, composite UB secondary $L/23$, RC column storey height/10 to /20. The default row per material is chosen automatically.
Loads	Slab self-weight from the system; EN 1991-1-1 (with its UK NA) for imposed loads and unit weights (concrete 25 kN/m³, softwood 4.2 kN/m³); combinations by EN 1990 expression 6.10 with the factors above.
Beams	Each beam (or run of beams joined by Continuous ends) is solved as a continuous beam by the shared beam FE solver, once for characteristic permanent and once for characteristic variable load, combined pointwise (6.10 envelope), with pattern loading. Secondary beams are solved first and their reactions are applied to primaries. Steel sections are the lightest Blue Book UB/UC (and other catalogue sections) meeting moment, shear and a deflection limit at the target utilisation.
Columns	Axial load only: beam reactions plus direct load plus self-weight over every level carried, no imposed-load reduction. Steel columns use an EN 1993-1-1 6.3.1 flexural buckling check at the storey height; RC columns are sized against a column capacity at the target.

Element	Basis
Footings	Bearing pressure on characteristic load, as 10.6.6.
Portals	As 10.6.5.
Carbon	Embodied carbon A1 to A3 from IStructE-based factors (concrete CEM II/A-L 217 $kgCO_2e/m^3$; reinforcement and steel from the IStructE default table; timber 0.25 $kgCO_2e/kg$ at 450 kg/m^3), with A4 and A5 allowances (default 12 % and 4 %). Reinforcement allowances: beams 130 kg/m^3, columns 160 kg/m^3, slabs per system. Intensity is divided by GIA and banded as SCORS. The carbon factors can be replaced by those chosen in the linked IFC node.

Member verdicts: each member gets a score against its target and is banded **Green / Amber / Red** with the same colours on plan, 3D and tables.

Lateral stability: Sketch-ET checks only that bracing or shear walls have been *placed* (and stop at the right levels), warning if a braced bay does not reach the foundation or stops below the top. The braces or walls, the wind and the notional horizontal loads are not designed.

10.9 Results

- **Plan and 3D** colour members by scheme colours (default), utilisation, material, "realistic", or none. Column reaction badges can show the ULS design value or the SLS characteristic $G_k + Q_k$ (switch on the View tab). Span arrows show slab spans; clicking one cycles the span direction.
- **Results** view: tables of slab panels, beams, columns, footings and portals, with verdicts.
- **Working** view: an audit sheet showing how each number was arrived at (floor build-up, beams, columns, footings, portal frames, load path).
- **Drawer**: Quantities (concrete, reinforcement, steel, timber), life-cycle modules, SCORS, carbon split by element and by material, and donuts.
- **Diagrams**: M, V and deflection diagrams per member, and moment envelopes for portal frames.

10.10 Site location and the Google 3D map

The **Site location (UK)** card (Building tab) places the model origin (0,0) on the OS National Grid and turns it so that the model's $+Y$ points on a bearing. It reuses the Wind designer's search and grid-reference code.

Input	Notes
Search box	Address, postcode (online lookup), place name, OS grid reference, or `lat, lon`. Suggestions float over the view in a card under the box.
Online lookup	Tick to use addresses and postcodes. Postcodes go to postcodes.io; addresses to Google Places when a Google key is connected, else Photon/OpenStreetMap; elevation to Google Elevation or Open-Meteo. Nothing leaves the browser until this or a map is switched on.
OS grid ref, E, N	Typed or pasted at any precision (for example `TL 04544 49099`). E and N are in metres. The grid-reference chip opens the 10/6-figure, E/N, lat/long and DMS card.
$+Y$	Degrees clockwise from grid north to the model's $+Y$ (0 = north-up).
Level	The ground floor (model level 0) in m AOD. Starts at the ground level read at the origin; type a level to raise or sink the building.
Refresh	Reads the ground level again.
Per-building names	Each template or frame group is listed with an editable name, its OS grid reference and the bearing its $+Y$ faces. Names show over the building in 3D.

Map in 3D buttons draw inside the 3D viewport: **Aerial** (Google satellite with a Google key, Esri World Imagery without one), **Street**, **Relief** (aerial on real ground heights from Google Elevation or Open-Meteo), and **3D** (the Google photorealistic 3D map). For the Aerial, Street and Relief maps a drop-down sets how much ground is drawn round the

model: 250 m, 500 m (default), 1 km or 2 km. On the Google 3D map the model is placed at its real member sizes: the 3D view is exported as a binary glTF and uploaded so the map can load it. Maps switch off when you leave Sketch-ET. The map views are views: switch them off to edit.

10.11 RIBA stages

Sketch-ET is a Stage 2 (Concept Design) tool, and its reports say so. The RIBA stage itself is a *project* setting in the project bar at the top of the workspace (default "Stage 2 - Concept Design"), not a Sketch-ET input; it stamps every report. Changing stage saves the stage you leave, and a new stage starts as a copy of the previous one; saved stages can be reopened. Sketch-ET's embodied-carbon totals are included in the project's RIBA stage carbon trend alongside the other nodes. Take the scheme into the detailed nodes (10.13) as the design progresses.

10.12 Reports and exports

Open **Export and IFC**:

- **Report**: choose paper size and orientation (default landscape), tick Plan image and 3D image, then **Word report (.doc)** or **PDF report** (opens the print dialogue; choose Save as PDF). The Plan or Split view must be open for the plan image, and the 3D or Split view for the 3D image.
- Report content (headline sections): summary rows (governing column N_{Ed} and size, worst beam, slab, characteristic loads, design load n, storeys, GIA, embodied carbon and SCORS band); Carbon Rating (SCORS); Preliminary layout (plan figure); 3D model as on screen; Elevation as on screen; portal frame figures; beam-type and footing-type figures; Mass by element; embodied carbon by element (A1 to A5) and by material (A1 to A3); **Preliminary status of this report** (the caveat above, what the analysis does and what it does not cover); Design inputs; Slab panels; Beam schedule (one row per size, worked for the governing member); Column schedule; Quantity take-off; Floor build-up working; Portal frames (analysis, sections, stability, working); Foundations schedule with the governing type worked; lightest and heaviest members; Warnings raised by the model; Notes on the method; and a Load path check for one floor.
- **Export IFC file**: IFC4 in metres, one element per member per level. It is a concept model of prismatic blocks at the serial sizes: a steel beam is a rectangle, not a rolled section; a slab is a flat plate. No connections, reinforcement, penetrations, fire protection or tolerances. Steel members carry their true kg/m so the receiving node does not misread mass. Coordinate with it; do not fabricate from it.
- **Send to the IFC node**: builds an IFC node from the scheme (a second send updates the same node). **Open the IFC node**, and **Pull carbon back**, which takes the A1 to A3 factors chosen in the IFC node (the EPDs) and applies them here so the SCORS band moves. This is the only two-way wire in the workspace.

10.13 Connections to other nodes

Each design exposes output dots: one for the design's **total** (sum of all column base reactions, G_k and Q_k) and one per column. The value sent is the characteristic base reaction as the `DLz` (G_k) and `LLz` (Q_k) pair in kN, with the design name, column section and verdict attached. It is axial only, with no frame moments and no imposed-load reduction.

To	Notes
Column, Foundation, Base plate	Column reactions feed the load sets.
Beam, Slab	Allowed wire; load transfer follows the receiving node's sync.
IFC	The scheme goes as a concept IFC model; EPD factors come back (two-way).
Maths	As a value source.

Sketch-ET has no inputs from other nodes. The Wind node does not connect to it; wind on a portal frame is typed on the frame's card.

10.14 Tips and limitations

- Calibrate first, then check a second dimension.
- Turn off **Square - 90 degrees only** when tracing an irregular existing building, and keep the master snap off to land the pointer exactly.
- Use the Lock > Frame switch while marking up so nothing moves by accident.
- Press Run on the node card before expecting values on its wires.
- The 14 m long-span flag is a prompt, not a limit.
- Slab thickness in the library is a recommendation from a rule; for composite decks the depth snaps to the profile's tabulated depths.
- Every storey is assumed identical; use per-column storey counts for a wing of different height.
- Deck and portal outputs depend on the Tata and SCI source data; check against the manufacturer's current tables before specifying.
- Browser storage limits apply to large PDFs because the sheet image is stored with the design.
- Known limits: no lateral design, no second-order effects, no punching shear, no connection or fire design.

> **References.** The standards, guidance and data sources used in this chapter are listed, with web links, in Chapter 17.

Screen reference - panel by panel

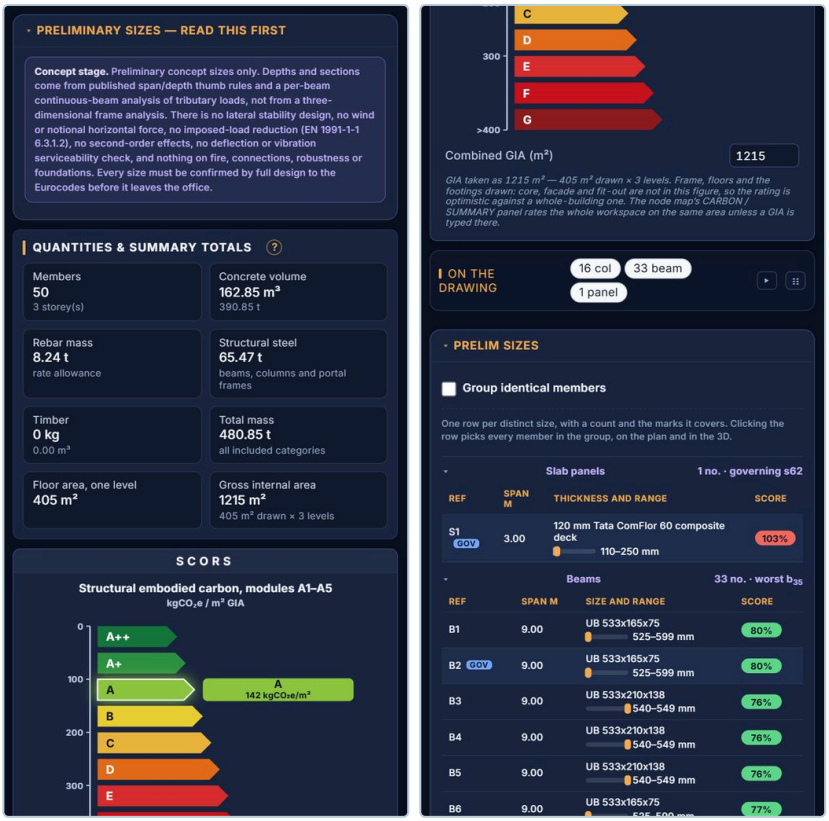
These screenshots show every panel of the module exactly as it appears on screen, scrolled from top to bottom, so you can match any field, button or result to the explanations in this chapter. Cards that are collapsed by default are shown expanded.

Plan view with a template loaded: the whole screen

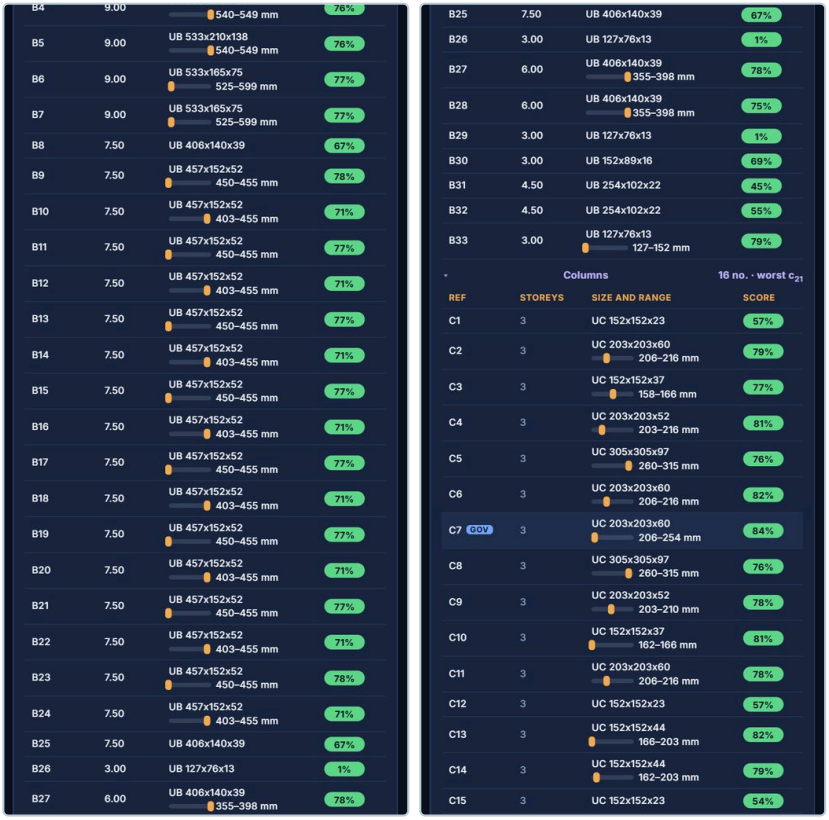


Plan view with a template loaded: the whole screen.

Plan view with a template loaded: results panel (right of the screen)



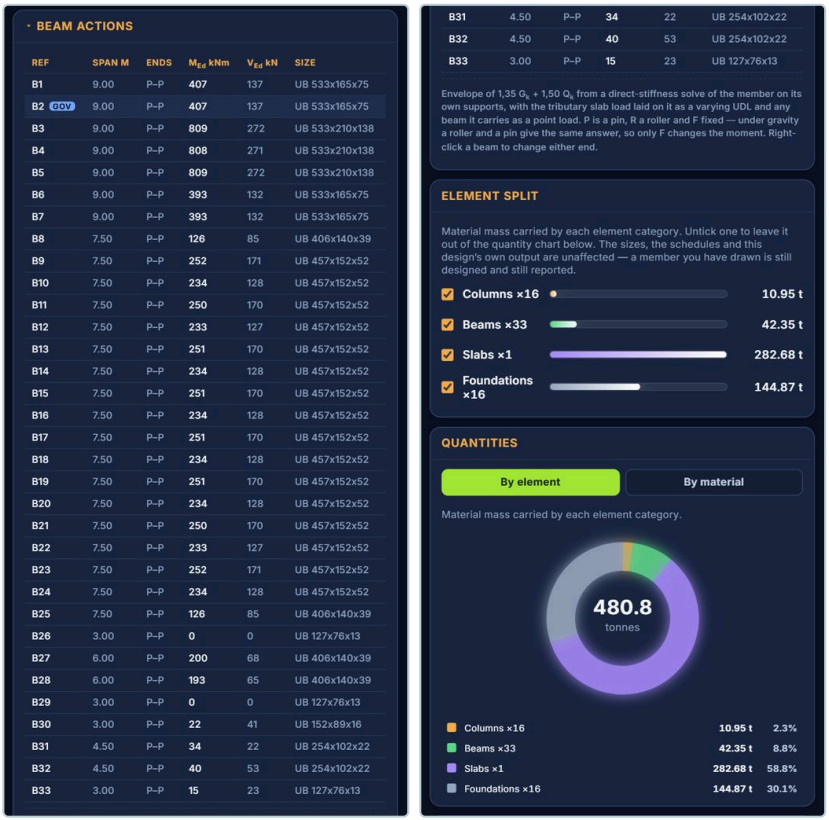
Plan view with a template loaded, results panel (right of the screen), parts 1-2 of 8. Cards shown: Preliminary sizes — read this first, Quantities & summary totals, Scors, On the drawing, Prelim sizes.



Plan view with a template loaded, results panel (right of the screen), parts 3-4 of 8.



Plan view with a template loaded, results panel (right of the screen), parts 5-6 of 8. Cards shown: Column reactions, Beam actions.



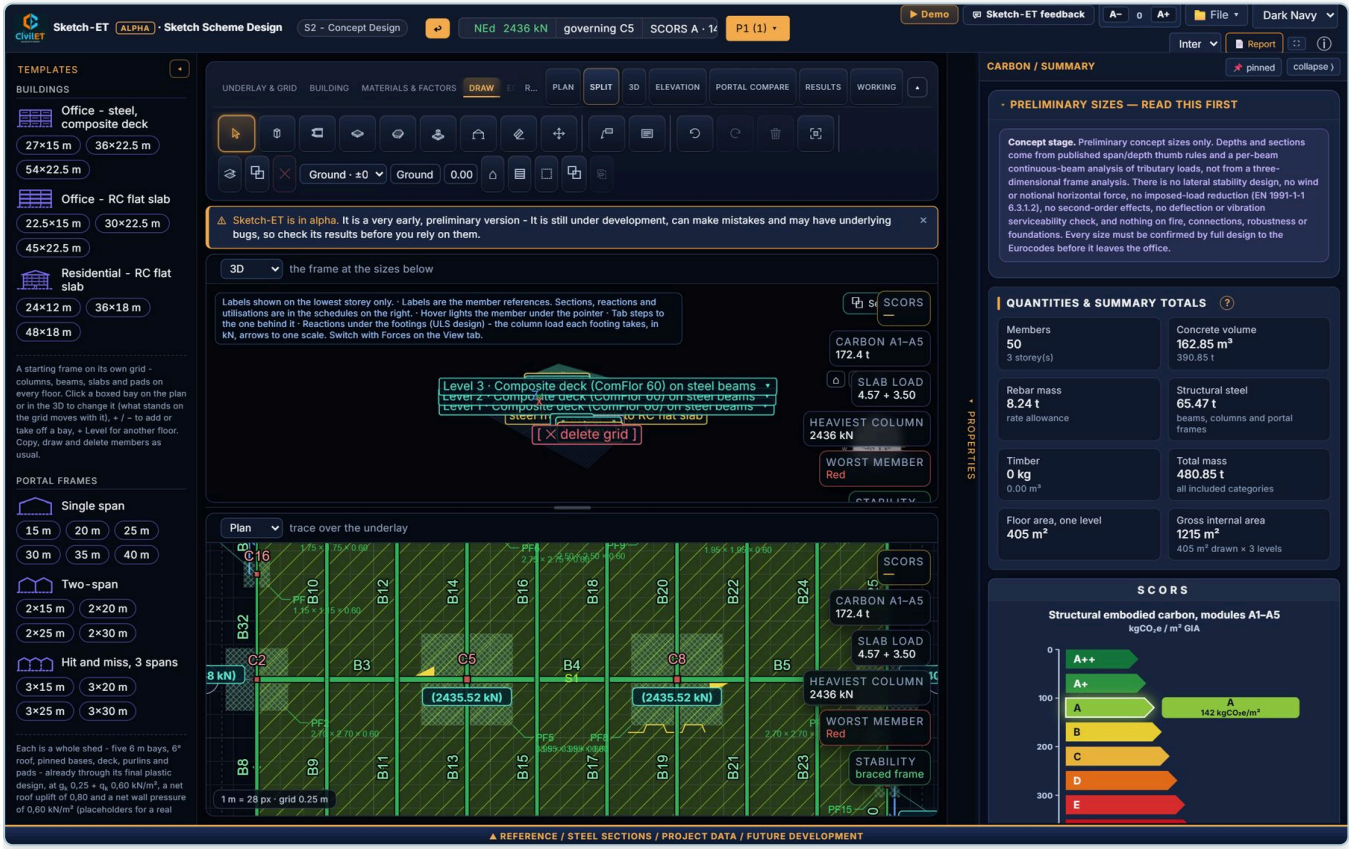
Plan view with a template loaded, results panel (right of the screen), parts 7-8 of 8. Cards shown: Beam actions, Element split, Quantities.

3D view: the whole screen



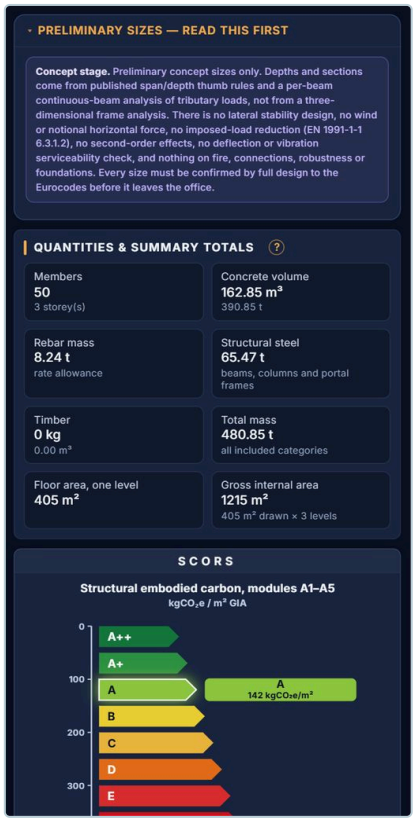
3D view: the whole screen.

Split view: the whole screen



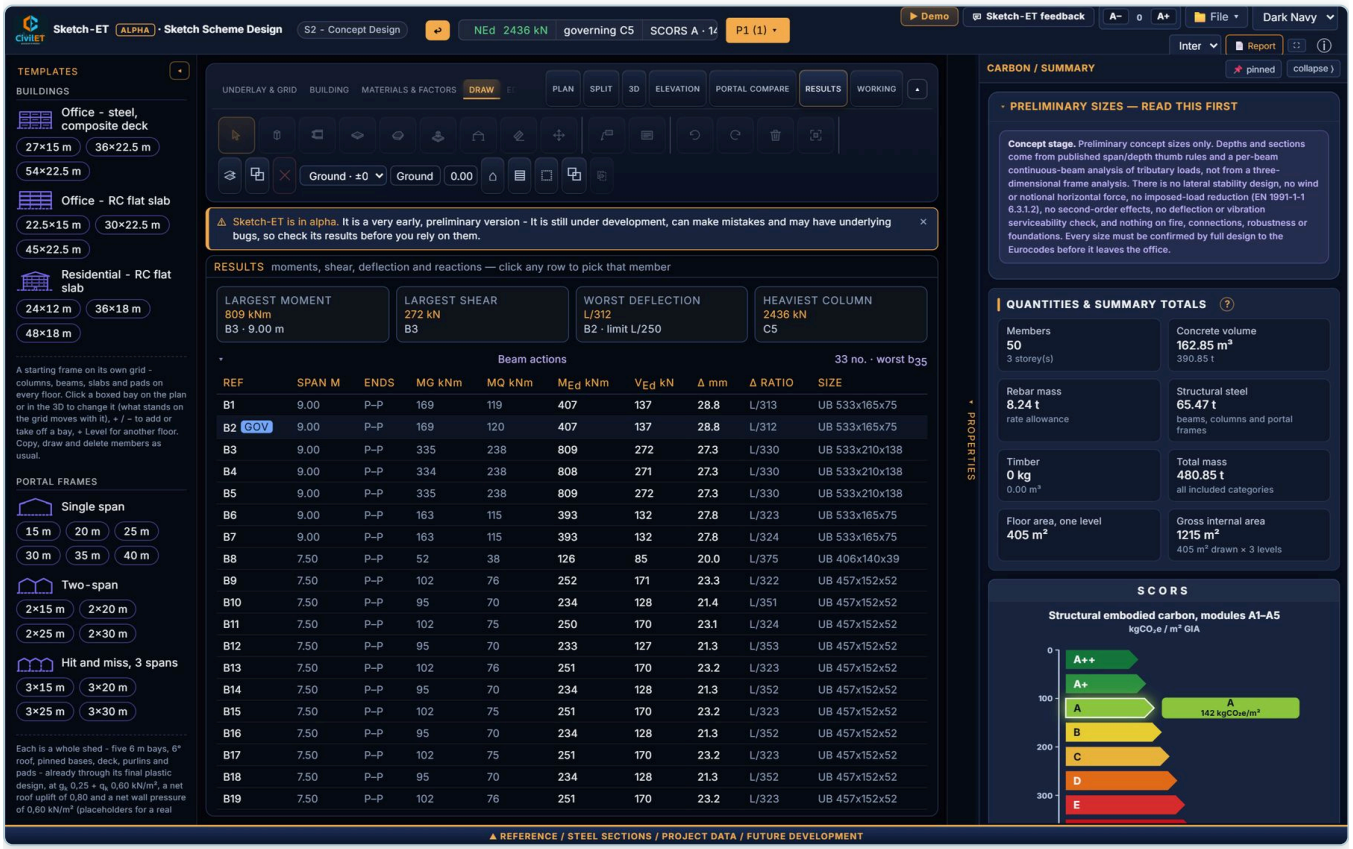
Split view: the whole screen.

Split view: results panel (right of the screen)



Split view, results panel (right of the screen), part 1 of 1. Cards shown: Preliminary sizes — read this first, Quantities & summary totals, Scors.

Results view: the whole screen



Results view: the whole screen.

Results view: working panel (centre of the screen)

LARGEST MOMENT	LARGEST SHEAR	WORST DEFLECTION	HEAVIEST COLUMN
809 kNm	272 kN	L/312	2436 kN
B3 · 9.00 m	B3	B2 · limit L/250	C5

Beam actions — 33 no. · worst b_{35}

REF	SPAN M	ENDS	MG kNm	MQ kNm	M_{Ed} kNm	V_{Ed} kN	Δ mm	Δ RATIO	SIZE
B1	9.00	P–P	169	119	407	137	28.8	L/313	UB 533x165x75
B2 GOV	9.00	P–P	169	120	407	137	28.8	L/312	UB 533x165x75
B3	9.00	P–P	335	238	809	272	27.3	L/330	UB 533x210x138
B4	9.00	P–P	334	238	808	271	27.3	L/330	UB 533x210x138
B5	9.00	P–P	335	238	809	272	27.3	L/330	UB 533x210x138
B6	9.00	P–P	163	115	393	132	27.8	L/323	UB 533x165x75
B7	9.00	P–P	163	115	393	132	27.8	L/324	UB 533x165x75
B8	7.50	P–P	52	38	126	85	20.0	L/375	UB 406x140x39
B9	7.50	P–P	102	76	252	171	23.3	L/322	UB 457x152x52
B10	7.50	P–P	95	70	234	128	21.4	L/351	UB 457x152x52
B11	7.50	P–P	102	75	250	170	23.1	L/324	UB 457x152x52
B12	7.50	P–P	95	70	233	127	21.3	L/353	UB 457x152x52
B13	7.50	P–P	102	76	251	170	23.2	L/323	UB 457x152x52
B14	7.50	P–P	95	70	234	128	21.3	L/352	UB 457x152x52
B15	7.50	P–P	102	75	251	170	23.2	L/323	UB 457x152x52
B16	7.50	P–P	95	70	234	128	21.3	L/352	UB 457x152x52
B17	7.50	P–P	102	75	251	170	23.2	L/323	UB 457x152x52
B18	7.50	P–P	95	70	234	128	21.3	L/352	UB 457x152x52
B19	7.50	P–P	102	76	251	170	23.2	L/323	UB 457x152x52

Results view, working panel (centre of the screen), part 1 of 4.

B18	7.50	P–P	95	70	234	128	21.3	L/352	UB 457x152x52
B19	7.50	P–P	102	76	251	170	23.2	L/323	UB 457x152x52
B20	7.50	P–P	95	70	234	128	21.3	L/352	UB 457x152x52
B21	7.50	P–P	102	75	250	170	23.1	L/324	UB 457x152x52
B22	7.50	P–P	95	70	233	127	21.3	L/353	UB 457x152x52
B23	7.50	P–P	102	76	252	171	23.3	L/322	UB 457x152x52
B24	7.50	P–P	95	70	234	128	21.4	L/351	UB 457x152x52
B25	7.50	P–P	52	38	126	85	20.0	L/375	UB 406x140x39
B26	3.00	P–P	0	0	0	0	0.1	L/22156	UB 127x76x13
B27	6.00	P–P	82	60	200	68	16.3	L/369	UB 406x140x39
B28	6.00	P–P	80	57	193	65	15.7	L/382	UB 406x140x39
B29	3.00	P–P	0	0	0	0	0.1	L/22156	UB 127x76x13
B30	3.00	P–P	9	7	22	41	8.3	L/363	UB 152x89x16
B31	4.50	P–P	14	10	34	22	8.1	L/558	UB 254x102x22
B32	4.50	P–P	16	12	40	53	9.9	L/454	UB 254x102x22
B33	3.00	P–P	6	5	15	23	9.5	L/315	UB 127x76x13

MG and MQ are the permanent and imposed halves of the same direct-stiffness solve; M_{Ed} is the 1,35 G_k + 1,50 Q_k envelope of it and V_{Ed} the shear from the same case. δ is the largest deflection under the CHARACTERISTIC load, off the same solve with each section's own E·I, checked against span/250. A concrete beam here is proportioned by a span/depth ratio instead, and says so rather than quoting a millimetre nobody calculated. Lateral-torsional buckling is NOT checked: full lateral restraint from the floor is assumed.

Column axial force — 16 no. · worst c_{21}

REF	STOREYS	G_k kN	Q_k kN	N_{Ed} kN	SIZE
C1	3	85.22	59.48	204.26	UC 152x152x23
C2	3	478.25	335.36	1148.68	UC 203x203x60
C3	3	198.02	135.93	471.23	UC 152x152x37

Results view, working panel (centre of the screen), part 2 of 4.



C2	3	478.25	335.36	1148.68	UC 203x203x60
C3	3	198.02	135.93	471.23	UC 152x152x37
C4	3	423.34	298.38	1019.07	UC 203x203x52
C5	3	1010.54	714.19	2435.52	UC 305x305x97
C6	3	494.88	345.32	1186.06	UC 203x203x60
C7	3	511.62	358.14	1227.91	UC 203x203x60
C8	3	1010.54	714.19	2435.52	UC 305x305x97
C9	3	409.39	287.69	984.20	UC 203x203x52
C10	3	209.17	144.37	498.93	UC 152x152x37
C11	3	474.91	332.81	1140.35	UC 203x203x60
C12	3	85.22	59.48	204.26	UC 152x152x23
C13	3	251.89	179.30	609.00	UC 152x152x44
C14	3	243.50	172.88	588.05	UC 152x152x44
C15	3	78.86	56.68	191.48	UC 152x152x23
C16	3	80.82	58.29	196.53	UC 152x152x23

Ultimate axial force at the BASE, EN 1990 (6.10), including the column's own weight over every level it carries and with no imposed-load reduction (EN 1991-1-1 6.3.1.2), which is conservative and increasingly so with storey count. Columns are axial-only: no frame moments are derived and no stability system is modelled.

Slab panels — 1 no. · governing s62

REF	SHORT M	LONG M	AREA m²	THICKNESS	SCORE
S1 GOV	3.00	7.50	405.0	120 mm Tata ComFlor 60 composite deck	103%

Foundations — 16 no. · all pass

REF	UNDER	NK kN	W kN	NK+W kN	SIZE	CHECK
PF1	C1	144.70	19.84	164.53	1.15 × 1.15 × 0.60 m pad footing	PASS 83%
PF2	C2	813.61	109.35	922.96	2.70 × 2.70 × 0.60 m pad footing	PASS 84%

Results view, working panel (centre of the screen), part 3 of 4.



REF	UNDER	NK kN	W kN	NK+W kN	SIZE	CHECK
PF1	C1	144.70	19.84	164.53	1.15 × 1.15 × 0.60 m pad footing	PASS 83%
PF2	C2	813.61	109.35	922.96	2.70 × 2.70 × 0.60 m pad footing	PASS 84%
PF3	C3	333.95	45.94	379.89	1.75 × 1.75 × 0.60 m pad footing	PASS 83%
PF4	C4	721.71	97.54	819.25	2.55 × 2.55 × 0.60 m pad footing	PASS 84%
PF5	C5	1724.73	234.04	1958.77	3.95 × 3.95 × 0.60 m pad footing	PASS 84%
PF6	C6	840.20	113.44	953.63	2.75 × 2.75 × 0.60 m pad footing	PASS 84%
PF7	C7	869.77	117.60	987.37	2.80 × 2.80 × 0.60 m pad footing	PASS 84%
PF8	C8	1724.73	234.04	1958.77	3.95 × 3.95 × 0.60 m pad footing	PASS 84%
PF9	C9	697.07	93.75	790.82	2.50 × 2.50 × 0.60 m pad footing	PASS 84%
PF10	C10	353.54	48.60	402.14	1.80 × 1.80 × 0.60 m pad footing	PASS 83%
PF11	C11	807.72	109.35	917.07	2.70 × 2.70 × 0.60 m pad footing	PASS 84%
PF12	C12	144.70	19.84	164.53	1.15 × 1.15 × 0.60 m pad footing	PASS 83%
PF13	C13	431.19	60.00	491.19	2.00 × 2.00 × 0.60 m pad footing	PASS 82%
PF14	C14	416.39	57.04	473.42	1.95 × 1.95 × 0.60 m pad footing	PASS 83%
PF15	C15	135.54	18.15	153.69	1.10 × 1.10 × 0.60 m pad footing	PASS 85%
PF16	C16	139.10	19.84	158.94	1.15 × 1.15 × 0.60 m pad footing	PASS 80%

Nk is the characteristic column load, $G_k + Q_k$. W is the footing's own weight — gross for a pad on a gross allowable pressure, net of the soil it displaces when the allowable is a net one — and Nk + W is what actually bears, which is the reaction to hand a foundation designer. Not factored, because that is what an allowable pressure is quoted against. No sliding, no overturning, no eccentricity, no punching, no bending, no reinforcement and no settlement.

Preliminary concept sizes only. Depths and sections come from published span/depth thumb rules and a per-beam continuous-beam analysis of tributary loads, not from a three-dimensional frame analysis. There is no lateral stability design, no wind or notional horizontal force, no imposed-load reduction (EN 1991-1-1 6.3.1.2), no second-order effects, no deflection or vibration serviceability check, and nothing on fire, connections, robustness or foundations. Every size must be confirmed by full design to the Eurocodes before it leaves the office.

Results view, working panel (centre of the screen), part 4 of 4.

CHAPTER 11

IFC Viewer and Embodied Carbon

11.1 Overview

The IFC viewer node turns a BIM export into three things:

1. A **quantity take-off**: elements counted and measured by type, material and storey.
2. An **embodied carbon assessment** (upfront carbon, modules A1 to A5) built on the IStructE guide *How to calculate embodied carbon*, 3rd edition (version 1.1, March 2025). Factors, transport scenarios, waste rates and site-activity rates are transcribed from its tables, and the table numbers are quoted in the interface.
3. A **3D viewer** where the model can be inspected, filtered, coloured by type, material or carbon, and individual members can be sent to the beam, slab, column or foundation design nodes.

The IFC node does not contain a design engine. When you design an element from the model, its geometry and loads are handed to a real design node over an ordinary wire (see 11.12).

All parsing is done in the browser. No file is uploaded anywhere.

11.2 Adding the node and loading a model

Step	What to do
Add the node	Open the node library on the node map and add **IFC viewer** (tag "BIM · quantities").
Load a file	Drop a `.ifc` file on the node's drop area, or click it to browse. The node's **Replace** button loads a different file.
Open the module	Press **Open IFC module** on the node card. This gives the full-screen module: inputs column on the left, 3D viewer with tabs in the middle, charts and reference tables in the bottom drawer, summary and SCORS rating on the right.
Quick look	Press **Open IFC 3D viewer** to dock the same viewer beside the node map without leaving the map.

Accepted input is a plain STEP physical file (`.ifc`). `.ifczip` , `.ifcxml` and `.zip` are refused with a message asking for a plain `.ifc` export.

If a file has no STEP entities the node reports "No STEP entities found - is this an IFC file?".

Several models

A workspace can hold several IFC nodes (one per model). The first IFC node is the *host*: further models are folded into its card. The card shows a **Models loaded** list with a tick for the model the card reports on (one at a time), a delete button for each, and **+ Add IFC file**, which creates a new IFC node and loads the file into it in one step. Each model keeps its own factors, ticks, GIA and element designs.

What is read from the file

Item	How it is read
Units	Length, area, volume and mass units are read from the project and normalised to m, m^2, m^3 and kg.
Storeys	`IFCBUILDINGSTOREY` name and elevation, sorted by elevation.
Elements	Every IFC class in the file is visited, not only the built-in list. Known classes go to a category (11.3). Unknown but physical classes are classified by name and listed in a warning so you can check them. Spatial and analysis classes (project, site, building, storey, space, grid, openings, structural analysis items, furniture, ports and similar) are never counted as elements.

Item	How it is read
Volume	The quantity-set volume is used when present (net volume first, then gross). Otherwise the volume is computed from the element geometry (extruded solids, faceted breps, polygonal face sets), and where openings are related to the element their volume is subtracted. The source is flagged on each element (quantity set, geometry, estimated geometry, or none).
Area, length, weight	From the quantity sets where present. A quantity-set weight is trusted only for an element whose material you have not re-assigned.
Material	From the element's material association, falling back to its type's. The name is matched to a material bucket (11.3).
Mesh	Solids are built for the first 1,600 elements for the 3D preview; the rest are counted in the quantities but omitted from the preview (a warning says so).

Special cases the loader handles, each with a warning in the node:

- **Revit fragments**: where an export gives a host's own net quantities to its fragments, each fragment is costed on its own share to avoid double counting.
- **Terrain meshes** are not counted as building material.
- **Assembly containers** (stairs, roofs, ramps) are skipped when their parts are counted individually.
- **Modelled reinforcement**: if the file contains explicit reinforcement elements, the rebar allowance is set to 0 for that model so reinforcement is not counted twice.
- **Large models**: above 4,000 elements the node keeps only category totals (the per-element list stays in memory for the session). Per-element re-assignment, the Elements list after a reload and per-element design are not available from totals alone. Re-load the file to work per element.

Persistence

Node state is stored in the browser. The 3D solids are stored with it when they are modest in size (about 700 kB of serialised mesh), and are also mirrored to the browser's database so large models reopen with their solids. If neither is possible, the node reopens with bounding boxes until the file is loaded again, and says so. State is per browser; moving to another machine means re-loading the file.

11.3 Categories and material buckets

Element categories

Category	IFC classes (built-in map)	Counted as structural by default
Columns	IfcColumn, IfcColumnStandardCase	Yes
Beams	IfcBeam, IfcBeamStandardCase, IfcMember, IfcMemberStandardCase	Yes
Slabs	IfcSlab (and standard/elemented cases)	Yes
Walls	IfcWall (and standard/elemented cases)	Yes
Piles	IfcPile	Yes
Pile caps	IfcFooting with predefined type PILE_CAP	Yes
Ground beams	IfcFooting with predefined type FOOTING_BEAM	Yes
Foundation slabs	foundation slabs recognised by the loader	Yes
Footings and pads	IfcFooting	Yes

Category	IFC classes (built-in map)	Counted as structural by default
Assemblies	IfcElementAssembly, IfcBuildingElementPart	Yes
Plates	IfcPlate (and standard case)	Yes
Stairs and ramps	IfcStair, IfcStairFlight, IfcRamp, IfcRampFlight	Yes
Roofs	IfcRoof	Yes
Braces and rods	IfcTendon, IfcReinforcingBar, IfcReinforcingMesh, IfcReinforcingElement	Yes
Other elements	curtain walls, coverings, railings, proxies, doors, windows, chimneys, shading devices, civil, geographic and transport elements	**No**
Manual additions	not an IFC class: your own top-ups (11.8)	Yes

Piles, pile caps and ground beams are separate line items because their reinforcement rates differ greatly.

On loading, categories flagged non-structural ("Other elements") are **ticked out** of the carbon totals once per node. Tick them back in the workspace *Element carbon split* list if you want them counted. Re-loading a file never reverts ticks you have made.

Material buckets

The material name (and the type name) is matched, in this order, to a bucket. The bucket decides the density used to turn volume into mass and the carbon factor applied.

Bucket	Default density	Matched by (summary)
Rebar	7,850 kg/m³	"rebar", or "reinforc..." where concrete or masonry words are absent
Masonry	1,800 kg/m³	masonry, brick, block, stone (tested before concrete, so "concrete block" is masonry)
Concrete	2,400 kg/m³	concrete, beton, C-grades such as C32/40, cement, "RC ", in situ, precast, screed, and coded names such as "-CON-"
Steel	7,850 kg/m³	steel, S235 to S450, metal, iron, and equivalents
Timber	450 kg/m³	timber, wood, glulam, CLT, LVL, softwood, hardwood
Aluminium	2,700 kg/m³	alumin...
Glass	2,500 kg/m³	glass, glazing
Other	2,000 kg/m³	any other named material
Unclassified	2,400 kg/m³	no material at all; guessed from the IFC class where possible (footing, pile, slab, wall as concrete; member, plate as steel; reinforcing, tendon as rebar), otherwise priced as "other"

"Reinforced concrete" is **concrete**: its reinforcement is modelled by the rebar allowance, not by the bucket. Concrete elements take the density of the factor data point they are priced on (11.5), not always 2,400 kg/m³.

11.4 The module screen

Inputs column (left)

A stack of collapsible, re-orderable panels (drag the handle to rearrange; open/closed state is remembered).

Panel	Purpose
Custom EPD factors	Store a manufacturer EPD reference file (PDF, XML or JSON, kept as a reference only) and enter a name and a factor for concrete ($kgCO_2e/m^3$), reinforcement, structural steel or timber ($kgCO_2e/kg$). The factor then appears as an extra bar on its chart and can be picked.
Element classification and rebar	Reinforcement allowance, per-category thumb rules, and re-classifying what a category counts as (11.6).
Carbon factors by element type	A different Table 2.5 data point per element type and material (11.5).
Manual additions	Quantities missing from the file (11.8).
IFC classes found	Read-only list of the IFC classes present with counts.
Take-off by element category	Read-only category table: number, m^3, tonnes, $kgCO_2e$.
A4 transport and A5 construction	Method, transport scenarios, A5 inputs (11.7).
Base reactions	Under development in this build (11.13).

The column can be collapsed to give the 3D stage the full width, and dragged to resize.

Viewer tabs (middle)

Tab	Content
Quantities	Model quantities of the included categories (elements, concrete m^3, rebar kg, structural steel, timber, masonry and other, total mass, A1 to A5 $kgCO_2e$) and the *By element type and material* table. Rows ticked out of the carbon split are dimmed and marked with a circled-slash symbol.
Per storey	Floor area per storey, level, slab count, measured area, a **Custom m^2** override box, volume and total elements per storey. Clicking a row isolates that storey. A button highlights the elements an area was measured from.
Elements	Searchable element list (name, material or type) with the effective type and material.
Working	The carbon calculation table: each quantity multiplied by its factor, then the A4 and A5 stage rows.
Design	Hand a picked member to a design node (11.12).
Reactions	Under development (11.13).
Comparison	Compare loaded models side by side (11.11).

Viewer toolbar

Control	Function
Fit	Fit the model in view (double-clicking the 3D view does the same).
Measure	Distance between two picked faces.
Area	Click points around a face and close the shape to measure an area.
Storey selector	Show one storey at a time.
Heat map	Off, total $kgCO_2e$, or $kgCO_2e$ per m^3 per element; with a scale of logarithmic, even spread by rank, or linear.

Control	Function
View style	Solid, Shaded + edges, Ghosted, White model.
Colour	By element type or by material.
Projection	Perspective or orthographic.
Shadows	Optional sun shadow (costs a second pass; leave off for big models).
Walk	Fly-through with W A S D, E and Q for up and down, Shift to run, Esc to exit.
Replace IFC	Load a different file for this node.
Compare models	Enter comparison (needs at least two loaded models).

Chips above the model show or hide each category, with sub-chips per material (click to toggle, double-click to isolate) and *Show all*. A 3D axis gizmo shows orientation. Large models are drawn as one merged mesh per category and storey, so orbiting stays smooth; picking still resolves to one element.

Right-click an element for its card: volume, areas, mass and carbon with the arithmetic shown, and the controls to re-assign it (11.6), mark it as a floor plate, or start a design from it.

Bottom drawer

- **Charts**: one comparison chart per material (11.5), with the lower, default and upper bound selector.
- **Reference**: the IStructE tables reproduced for checking (Table 2.5 factors, 2.6 transport, 2.7 default A4, 2.9 waste, 2.13 and 2.14).

Right panel

Quantities and Summary Totals, *Life cycle modules A1 to A5*, the bounded range card, *Element carbon split* (the include/exclude ticks), and the **SCORS** rating (11.9).

11.5 A1 to A3: product stage factors

Method

For each element, the module prices the material mass or volume against a factor:

- **Concrete**: $\text{A1-A3} = \text{volume}\ (\text{m}^3) \times \text{factor}\ (\text{kgCO}_2\text{e}/\text{m}^3)$. The per-m³ factor is derived as the guide's kgCO₂e/kg value multiplied by that data point's assumed density.
- **All other materials**: $\text{A1-A3} = \text{mass}\ (\text{kg}) \times \text{factor}\ (\text{kgCO}_2\text{e}/\text{kg})$.
- **Concrete reinforcement** is priced separately: allowance (kg per m³ of concrete) × concrete volume × the rebar factor. It is a different product from the concrete and is delivered, wasted and disposed of on the rebar line.

Which factor applies (precedence)

1. The element's own pick (right-click card, "Factor:" selector).
2. The pick for its element type and material (inputs column, *Carbon factors by element type*, or the **Applies to** control on a chart).
3. The model-wide pick for that material (the default selection for each material below, or the typed number).

A pick is either an IStructE Table 2.5 data point (which carries its own published lower and upper bounds) or a custom EPD (which publishes no range). A pick made for one material is ignored if the element is later re-bucketed to another, so steel is never priced off a concrete data point.

Default data points

Each material is initialised to a Table 2.5 data point at its recommended default. The typed value mirrors the current bound.

Material	Default data point	Value used
Concrete	UK in situ C32/40 (25% GGBS)	271.2 $kgCO_2e/m^3$ (0.113 $kgCO_2e/kg \times 2{,}400\ kg/m^3$)
Reinforcement	UK reinforcement bar (consumption average)	0.72 $kgCO_2e/kg$
Structural steel	UK rolled open sections (consumption average)	1.64 $kgCO_2e/kg$
Timber	UK softwood, import weighted average	0.156 $kgCO_2e/kg$
Masonry	UK dense concrete blocks	0.085 $kgCO_2e/kg$
Aluminium	Aluminium sheet, European consumption	6.58 $kgCO_2e/kg$
Glass	Glass, general	1.44 $kgCO_2e/kg$
Other	none (own factor)	0 until you enter one

Other presets available include UK in situ C8/10 to C40/50, global average C32/40, UK precast C40/50, mortar and screed, PT strand, stainless rebar, closed sections, plate, galvanised sheet, reused steel, CLT, glulam, LVL, I-joists, plywood, OSB, AAC and clay brick, aluminium extrusions, toughened glass, and an "other" list (aggregate, stone, plasterboard, asphalt, bitumen, intumescent coatings, cementitious fire spray, Portland cement). Concrete presets carry the density used to convert the factor and to mass the concrete: for example the screed/mortar row is 2,000 kg/m^3 and UK precast C40/50 is 2,385 kg/m^3. The mass follows the same data point, because A4, C2, C3-C4 and A5.3 are charged on mass.

Lower, default, upper bound

The **Lower / Default / Upper** selector on the Charts tab re-prices the whole model at the guide's typical lower, recommended default or typical upper value for every material. The bounds are the spread of products a project might buy, not error bars on one product. The guide advises reporting the default and quoting the range. The right panel and report show the three totals and the percentage spread when a published range exists for at least one factor in use. With only custom factors there is no range and the three bounds collapse onto one value (the module says so).

Charts

One chart per material shows the main Table 2.5 data points for that material (a subset such as C20/25 to C40/50, global and precast for concrete; not every data point has a bar, the rest are reachable as custom EPD factors). Clicking a bar writes that data point into the factor for the material (or for the element type chosen under **Applies to**). Bars carry a whisker for the data point's own published range. Custom EPDs add bars. Chips choose which materials' charts are shown. Materials absent from the take-off are marked and can still be switched on for comparison.

11.6 Reclassification and reinforcement allowance

Reinforcement allowance

For concrete without modelled reinforcement, a rebar mass is added: $\text{volume} \times \text{rate}$. The node-wide default is **110 kg/m^3**.

Option	Behaviour
Node-wide allowance	One rate for every cubic metre of concrete without modelled rebar.
Thumb-rule picker	Sets the node-wide rate to the middle of a published range for a chosen element type (bases 90 to 130, beams 250 to 350, columns 200 to 450, plate slabs 95 to 135, two-way slabs 67 to 135, walls 70 to 100, and so on).

Option	Behaviour
Per-category thumb rules (tick)	Each category uses the published average for its type: columns "Columns", beams "Beams", slabs "Slabs (two-way)", walls "Walls (normal)", foundations "Footings", stairs "Stairs", roofs "Slabs (one-way)", plates "Plate slabs", piles "Columns", pile caps "Pile caps", ground beams "Ground beams", foundation slabs "Rafts". The mid value of the range is used.
Typed per-category rate	A rate typed in the classification table beats the thumb rules. Blank uses the node-wide allowance (or the thumb rule if ticked).
Modelled reinforcement	If the model holds reinforcing elements, every rate is 0.

The thumb-rule table is reproduced from the One Click LCA help article "Average quantities of reinforcement in concrete" (UK table, attributed there to Reinforcement Estimates) and the node links to it. The node cannot judge your actual reinforcement: treat the allowance as a stand-in until bar schedules exist.

Re-classifying categories and elements

The exporter's class and material are a starting point. Three layers can correct them, in this precedence: this element, then its category, then the file.

- **Category level** (classification table): set "Counted as" to concrete, steel, rebar, timber, masonry, aluminium, glass or other for a whole category (for example "Other elements" that are really concrete). Changing it re-buckets that category everywhere: take-off, charts and carbon.
- **Element level** (right-click card): change the **Type** (moves it to another category and takes that category's rebar rate with it), the **Material**, and the **Factor** data point. Overrides are keyed to the IFC entity id, so they survive a reload of the same file. Loading a *different* file clears them (a warning says how many). The controls are disabled for models stored as category totals only.
- An override that restates what the file already says, or "auto", removes the override.

Reclassifying changes density (mass follows the new material), the factor, and the rebar allowance.

11.7 A4 and A5: transport and construction

Calculation method

Method	Behaviour
IStructE per-material (default)	A4 from Table 2.7 per material; A5 split into A5.1 to A5.4.
Percentage uplift (legacy)	A4 = **12%** and A5 = **4%** of A1-A3 (both editable). Described in the interface as a stand-in, not the IStructE method.

A4 transport to site (Table 2.7)

$\mathrm{A4} = \text{delivered mass} \times \mathrm{ECF}_{A4}$ ($kgCO_2e/kg$), by transport scenario per material. Concrete is charged on its mass (excluding rebar), and the rebar allowance on its own line.

Scenario	Road (km)	Sea (km)	$kgCO_2e/kg$	Typical use
Locally manufactured, ready-mixed concrete	20	0	0.003	Ready-mixed concrete
Locally manufactured (general)	50	0	0.009	Aggregate, earth, asphalt
Regionally manufactured	80	0	0.014	Plasterboard, blockwork, brick
Nationally manufactured	120	0	0.021	Structural timber, UK steel, precast
UK steel, 50% national / 50% European	810	50	0.1415	UK structural steel and reinforcement

Scenario	Road (km)	Sea (km)	kgCO₂e/kg	Typical use
European manufactured	1,500	100	0.262	CLT, glulam, European steel, PT strand
Globally manufactured	500	10,000	0.284	Specialist stone, bamboo
Custom	-	-	typed	Project-specific logistics

Default scenario per material: concrete ready-mixed; rebar and steel "UK steel"; timber national; masonry regional; aluminium and glass European; other regional. Distances include the UK average 30% empty running on the return road leg (a return-leg factor of 0.43 of the outbound distance, section 2.2.3 of the guide). The Table 2.6 transport emission factors are reproduced in the reference tables.

A5 construction process

Module	Basis	Default
A5.1 pre-construction demolition	demolished area (m²) × rate	**Area 0** (leave at 0 for a clear site; it is the existing floor area being demolished, not the new GIA). Rate 17.5 kgCO₂e/m² (50% of the RICS all-discipline 35, attributed to structures).
A5.2 construction activities	GIA (m²) × rate	Area = this model's GIA (set on the SCORS card). Rate 20 kgCO₂e/m² (50% of the RICS 40).
A5.3 material waste on site	$\mathrm{WF} \times (\mathrm{A1\text{-}A3} + \mathrm{mass} \times (\mathrm{A4} + \mathrm{C2} + \mathrm{C3\text{-}C4}))$ per material (Eqn 2.4)	Waste factors and C3-C4 factors below.
A5.4 worker transport	entered value, kgCO₂e	**0**; optional. Enter only the worker transport arising from the elements in scope.

Waste factors $\mathrm{WF} = 1/(1 - \mathrm{WR}) - 1$ (Table 2.9) used by default, per material: concrete 0.053, rebar 0.053, steel 0.010, timber 0.020, masonry 0.064, aluminium 0.010, glass 0.010, other 0.042. C2 (transport of waste away from site, Table 2.13) defaults to 0.009 kgCO₂e/kg. C3-C4 (Table 2.14 example values) defaults: concrete 0.0012, rebar 0.0013, steel 0.0013, timber 0.013, masonry 0.0012, aluminium 0.0013, glass 0.013, other 0.013 kgCO₂e/kg. The full Table 2.9 waste list is in the reference tables.

The panel is **locked** to the published values. Press the lock button to customise any rate (project logistics, contractor waste data) and press again to return every rate to the published value. A0 is taken as zero for buildings (the module says so).

A5.1, A5.2 and A5.4 are project-level figures and belong to no single element. They are apportioned pro rata on A1-A3 only so that the element splits add up to the headline. An individual element's carbon in the inspector is therefore A1-A3 + A4 + A5.3 only.

Total

$$\mathrm{A1\text{-}A5} = \mathrm{A1\text{-}A3} + \mathrm{A4} + \mathrm{A5}$$

The node card, right panel, charts and report all use this total.

11.8 Manual additions

For quantities missing from the file, enter: **Concrete m³, Rebar kg, Steel kg, Timber kg, Masonry kg**. They appear as a separate category called "Manual additions" so they stay auditable and can be ticked in or out on their own. They count as items, not as elements (so they cannot inflate the element count). Manual concrete takes its mass from the concrete density in force and gets **no** rebar allowance on top, since you have a rebar box of your own. Manual additions get their own row in the element-type factor table.

11.9 Workspace carbon summary and SCORS rating

Contribution to the workspace

Each IFC node contributes to the workspace embodied carbon list one row per **element type and material** (for example "Columns · Steel" separate from "Columns · Concrete"), with A1-A5 broken down per material. These rows carry include/exclude ticks in the *Element carbon split* list. The IFC total sits beside foundation, column, base plate, beam, slab and Sketch-ET totals in the workspace summary and in the RIBA stage comparison. A tick on a whole type, or on one type and material, removes it from the card, the right panel, charts and report alike. Partly excluded types are marked with a half-filled symbol.

SCORS rating

The right panel shows the SCORS rating, where the intensity is the A1-A5 carbon divided by GIA:

$$\text{intensity} = \frac{\text{A1-A5 } (\text{kgCO}_2\text{e})}{\text{GIA } (\text{m}^2)}$$

Band	A1-A5 intensity ($kgCO_2e/m^2$)
A++	0 to 50
A+	50 to 100
A	100 to 150
B	150 to 200
C	200 to 250
D	250 to 300
E	300 to 350
F	350 to 400
G	over 400

GIA can be typed on the card. Each IFC model carries its own GIA; if none is set, the model inherits the workspace figure. The workspace figure is the typed value if one was entered, otherwise the sum of the Sketch-ET designs' own GIA, otherwise a fallback of 1,000 m^2 (the panel says when the figure is derived or fallen back to). On the **Per storey** tab, the **Use … m^2 as this model's GIA** button writes the summed storey floor areas to the model. Floor area per storey comes from marked floor plates if any are marked on that storey, otherwise from that storey's slabs (using the quantity-set area, else the plan footprint of the element). A **Custom m^2** value overrides a storey; the measured figure stays visible beside it. Changing GIA changes A5.2, since A5.2 is charged on GIA.

11.10 Outputs, ports and wiring

Quantity output ports

The node card exposes these scalar outputs (per model):

Port	Value	Unit
Concrete volume	included concrete volume	m^3
Steel mass	included structural steel mass	kg
Rebar (est.)	included reinforcement mass	kg
Timber mass	included timber mass	kg
Total mass	included total mass	kg
A1-A5 CO_2e	included upfront carbon	$kgCO_2e$

These can be wired to a Math node (IFC outputs may feed a Math node input). Totals honour the include/exclude ticks.

Design ports

One output dot appears for each element for which a design has been started (11.12). The wire goes into the matching beam, slab, column or foundation node.

Sketch-ET link

A Sketch-ET (preliminary design) node can send its scheme to an IFC node as a concept model (the wire has a head at both ends; only the Sketch-ET to IFC direction is stored). A card built that way shows a **Back to Sketch-ET** button.

11.11 Comparing models

With two or more models loaded, **Compare models** puts up to five side by side, each in its own 3D pane with its own metrics: A1-A5, intensity per GIA, GIA (own or workspace), total mass, concrete volume, steel and reinforcement mass, and the bound and method used. Charts and a percentage spread (highest against lowest) are shown under the panes. If the models are not costed on the same bound and method, the report says that part of the difference is that setting rather than the design. Because the panes hold a snapshot of the metrics, editing an input rebuilds the comparison.

11.12 Designing from the model

Right-click a beam, slab, column, pad, pile cap or similar member and choose **Design** (or use the Design tab). The Design tab:

- Reads the member's geometry from the model.
- Asks for area loads (g_k and q_k, in kN/m^2) and, for columns and foundations, the number of floors.
- Hands geometry and loads to a **real beam, slab, column or foundation node** over a wire. Every check, carbon number and report line is then that module's own.

Mapping from category to design kind: beam and ground beam to Beam; slab and foundation slab to Slab; column and pile to Column; footing/pad and pile cap to Foundation. The tributary rule is deliberately simple: a panel is bounded by the nearest parallel members either side, treated as a two-way slab shedding at 45° (a trapezoid on the long edge and a triangle on the short edge).

Constraints: a design point driven from the IFC model is locked against other incoming wires (the other node's load row shows as model-owned); remove the design link in the IFC Design tab first. Typing your own loads on the receiving node sets an "IFC loads override" flag so the model no longer owns them. A design whose element is no longer in the loaded model is shown as a stub that can be removed. Designs are listed on the Design tab with utilisation or "not wired" or "out of date".

11.13 Known limitations

- **Base reactions / Reactions tab**: the take-down (applied loads, method, load panels, ULS combination) is marked *Under development* in this build and is not released. Do not use it for design.
- Only the upfront modules **A1 to A5** are calculated. C2 and C3-C4 are used only inside the A5.3 waste term; they are not reported as separate life cycle results, and no B or D modules are calculated.
- Factors are the guide's UK data points; the other preset families (global, European) are for comparison. Check the table against the book for any figure you rely on.
- The element-level carbon excludes the area-based A5.1, A5.2 and A5.4 (11.7).
- Compressed or XML IFC files are not read.
- Over 4,000 elements the model is held as category totals (11.2).
- Only the first 1,600 elements get 3D solids built from geometry.
- The rebar allowance is a thumb-rule estimate, not a design quantity.
- The report image of the 3D view is captured only if the viewer is open at the time.

11.14 Report

Open the IFC module and press the header **Report** button. The report is built by the shared module-report builder (same layout as the other modules) with these specifics:

- Title "IFC MODEL - QUANTITIES AND EMBODIED CARBON". It is a take-off, so there is no pass/fail banner.
- Opens with the **SCORS** rating graph (only if a GIA is set; without one no rating is fabricated) and four pies: mass by category, mass by material, A1-A5 by category, A1-A5 by material.
- Metric strip, model source (file name, schema, elements included, storeys, class count), material take-off, per-category and element-type-by-material tables.
- *Embodied Carbon* chapter: A1-A3, A4, A5.1 to A5.4, A5 total and A1-A5 total with the basis of each, lower and upper bound totals and range (when a published range exists), plus the life cycle module bars, bounded range bars and element carbon split bars.
- Calculation tables: carbon working, A4 per material with its mass, A5.3 per material, the classification and rebar table, manual additions, IFC classes found, and (if comparing) a model comparison table.
- Figures: a captured 3D view and one Table 2.5 chart per material shown on the Charts tab.

The report is live: if the inputs change while it is open, it re-reads the take-off. Excluded categories are listed but flagged and not in the totals.

11.15 Tips

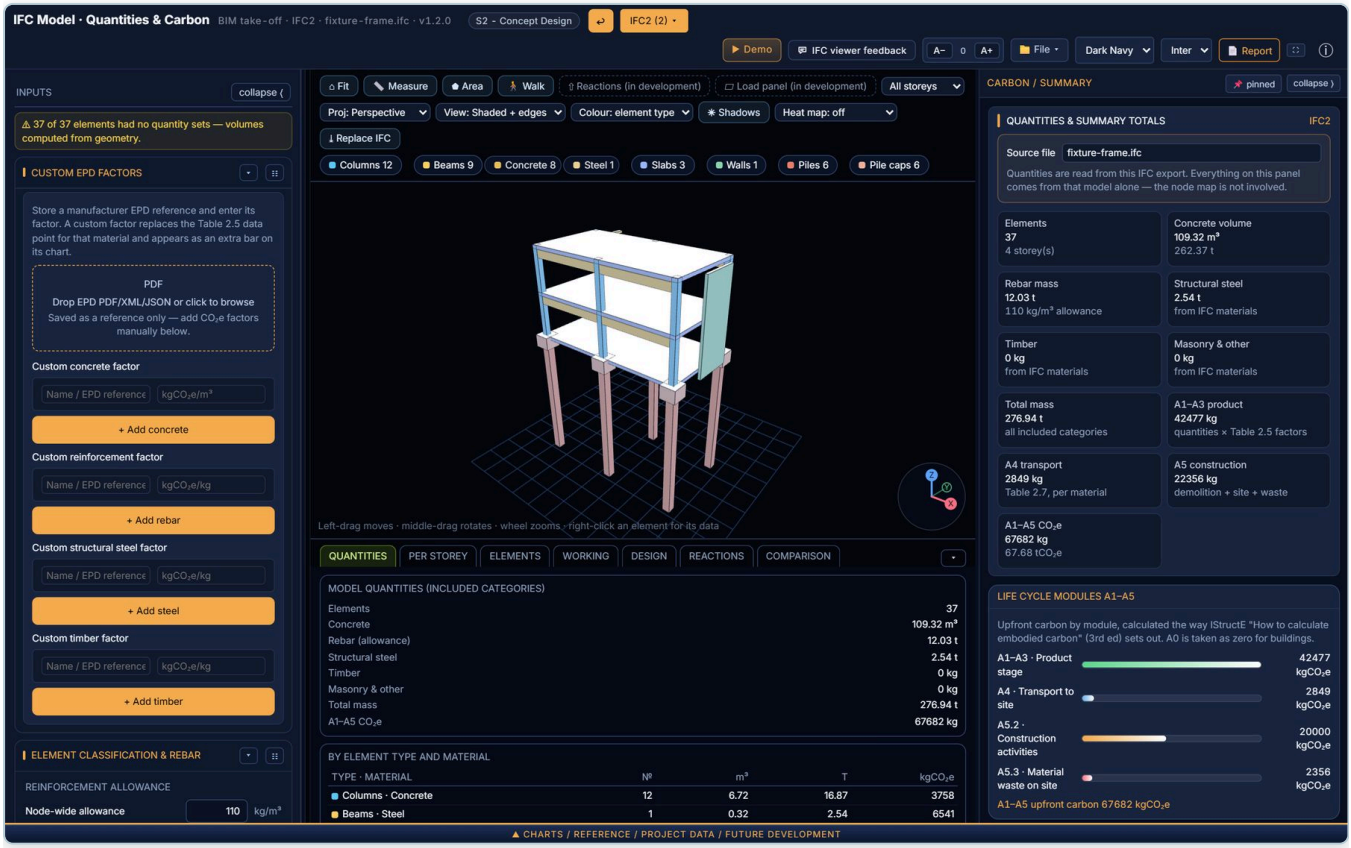
- Check the **warnings** under the file chip first. They list classes classified by name, geometry-derived volumes, fragments and omitted solids.
- If "Other elements" or a coded material shows up as concrete or steel in the wrong place, fix it by category in *Element classification and rebar*; use the right-click card only for single odd members.
- Report the **default** and quote the lower to upper range.
- Set GIA before quoting A5.2 or a SCORS band. Leave A5.1 area at 0 for a clear site.
- Unlock the A4 and A5 panel only when you have real logistics or contractor waste data.
- Non-structural categories start excluded; tick them in if the scope includes them.

> **References.** The standards, guidance and data sources used in this chapter are listed, with web links, in Chapter 17.

Screen reference - panel by panel

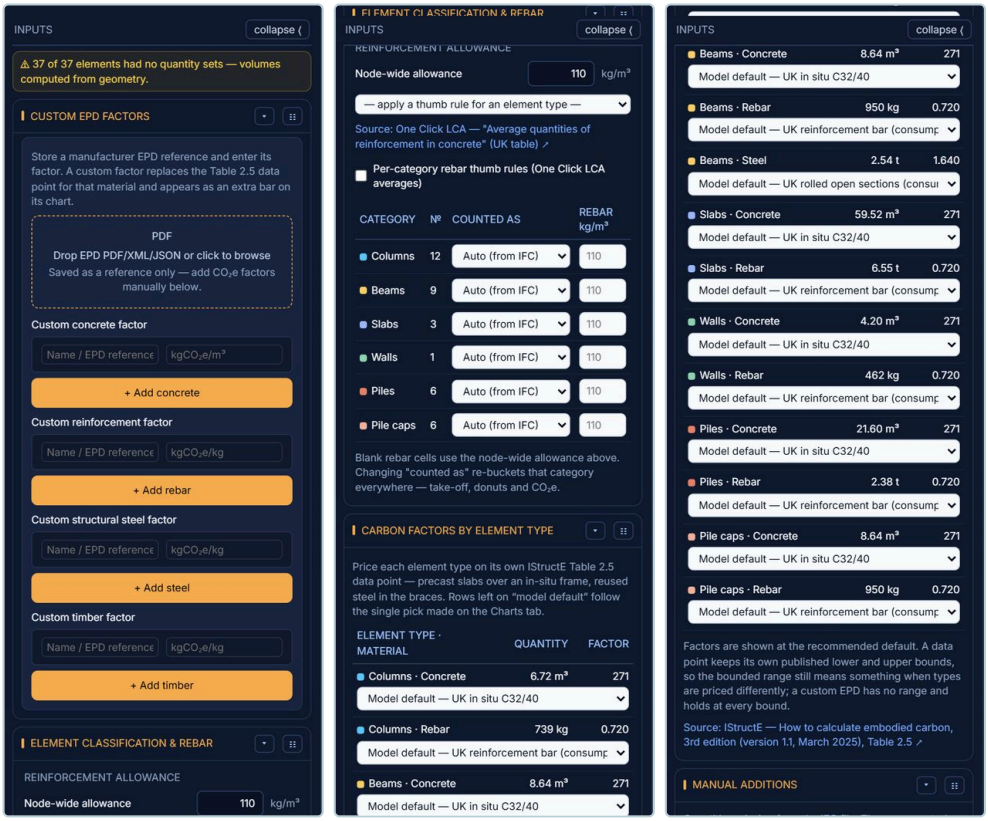
These screenshots show every panel of the module exactly as it appears on screen, scrolled from top to bottom, so you can match any field, button or result to the explanations in this chapter. Cards that are collapsed by default are shown expanded.

IFC module with a sample model: the whole screen

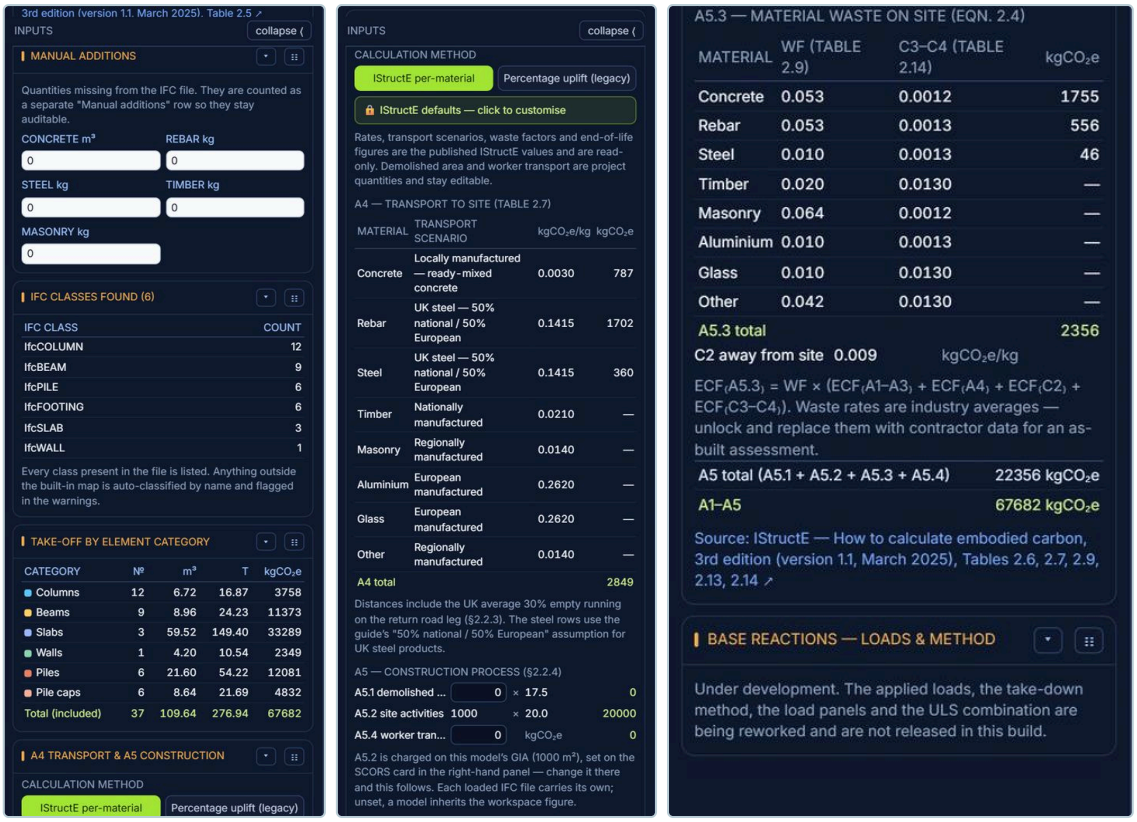


IFC module with a sample model: the whole screen.

IFC module with a sample model: input panel (left of the screen)

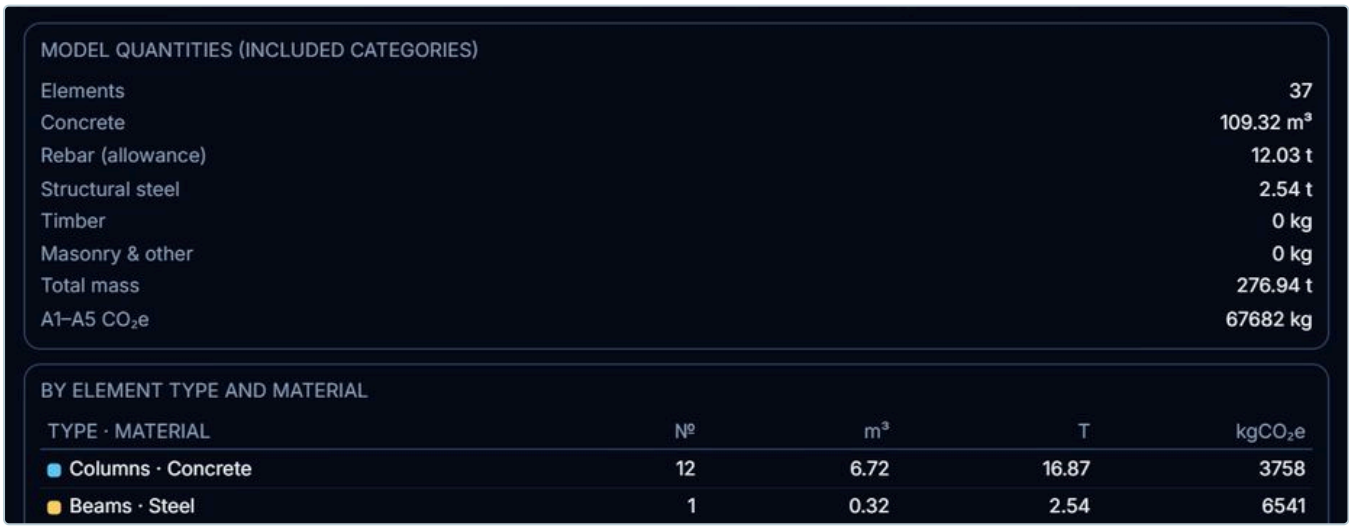


IFC module with a sample model, input panel (left of the screen), parts 1-3 of 6. Cards shown: Custom EPD factors, Element classification & rebar, Reinforcement allowance, Carbon factors by element type, Manual additions.

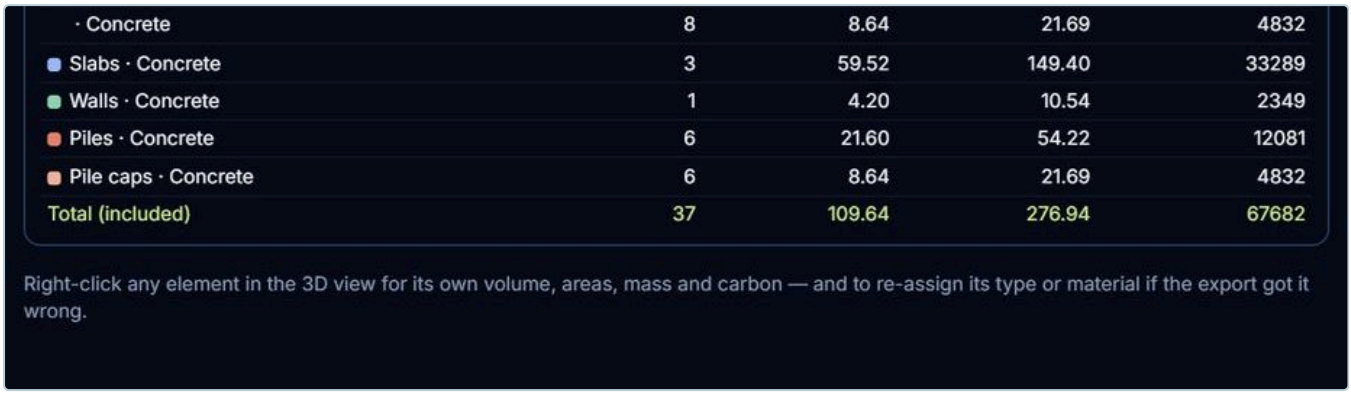


IFC module with a sample model, input panel (left of the screen), parts 4-6 of 6. Cards shown: Manual additions, IFC classes found (6), Take-off by element category, Calculation method, Base reactions — loads & method.

IFC module with a sample model: working panel (centre of the screen)



IFC module with a sample model, working panel (centre of the screen), part 1 of 2. Cards shown: Model quantities (included categories), By element type and material.

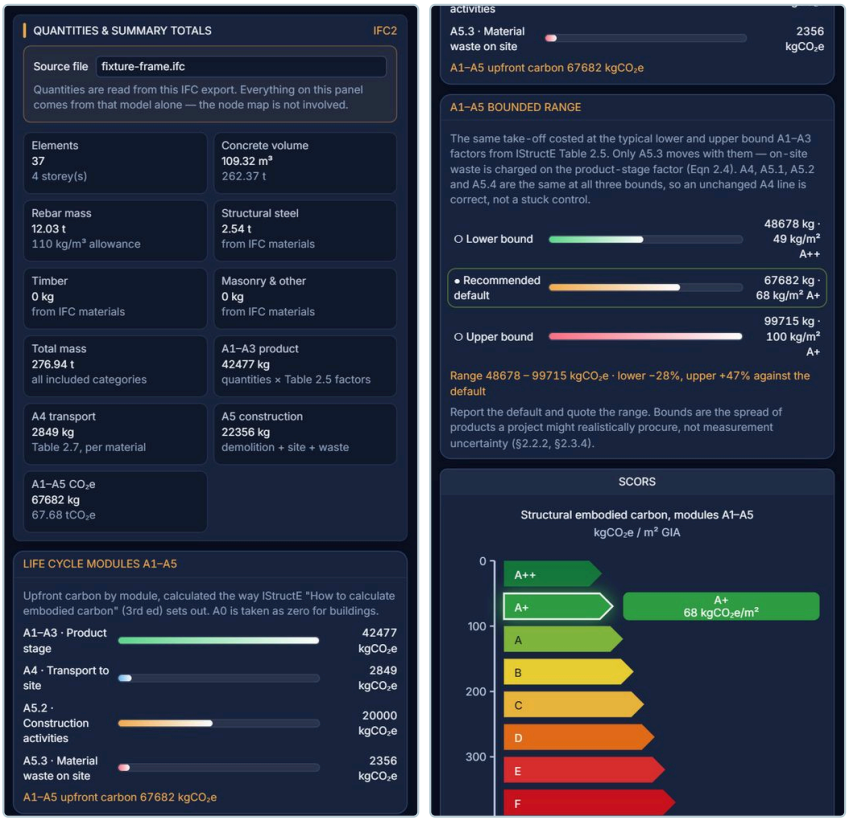


· Concrete	8	8.64	21.69	4832
Slabs · Concrete	3	59.52	149.40	33289
Walls · Concrete	1	4.20	10.54	2349
Piles · Concrete	6	21.60	54.22	12081
Pile caps · Concrete	6	8.64	21.69	4832
Total (included)	37	109.64	276.94	67682

Right-click any element in the 3D view for its own volume, areas, mass and carbon — and to re-assign its type or material if the export got it wrong.

IFC module with a sample model, working panel (centre of the screen), part 2 of 2.

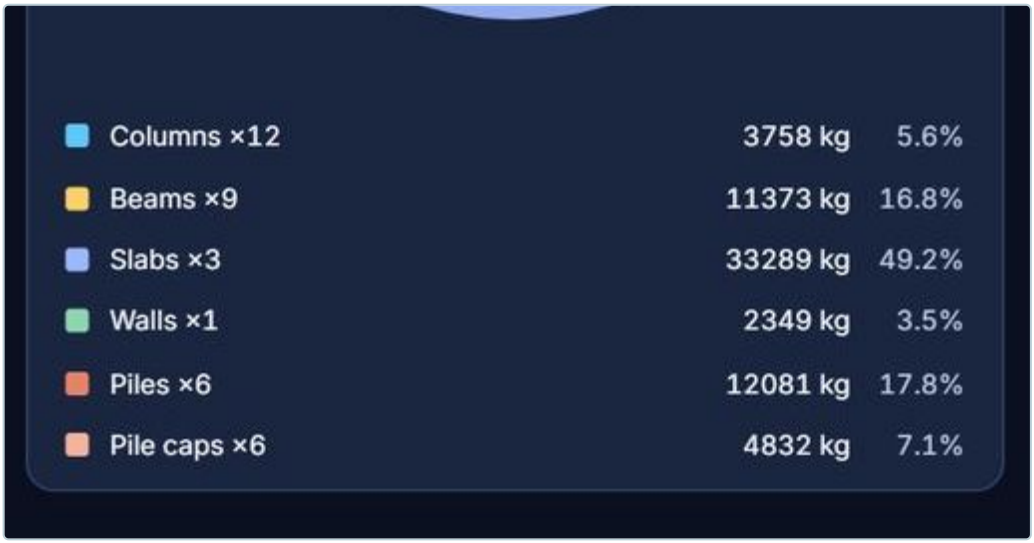
IFC module with a sample model: results panel (right of the screen)



IFC module with a sample model, results panel (right of the screen), parts 1-2 of 5. Cards shown: Quantities & summary totals IFC2, Scors.



IFC module with a sample model, results panel (right of the screen), parts 3-4 of 5. Cards shown: Element carbon split, Quantities.



IFC module with a sample model, results panel (right of the screen), part 5 of 5.

CHAPTER 12

Cut and Fill

12.1 Overview

The Cut & Fill node estimates earthworks volumes. You give it a topographic survey as an ASCII DXF file, draw one or more excavation outlines on it, set a formation level for each, and it reports **cut**, **fill** and **net** volumes. It also draws the ground as a triangulated surface, contours, plan, 3D view and section lines.

Status: alpha. The module carries an "Alpha" badge and a warning note ("a very early, preliminary version … can make mistakes and may have underlying bugs"). Check results independently before relying on them. The module is not a design check. It produces a take-off (the report says "TAKE-OFF", with no pass/fail banner).

What it does:

1. Reads levels from a DXF survey: spot heights as points, block attributes, text labels next to points, and contour polylines.
2. Builds a ground model: a triangulated irregular network (TIN) of the levelled points.
3. Lets you draw excavation shapes (or type a footprint) and set a formation level.
4. Integrates the volume between the ground and the formation, triangle by triangle.
5. Optionally places a building from an IFC model on the survey and digs for its buried elements.
6. Produces a report with the plan, 3D view and sections.

What it does not do: it does not apply bulking or compaction. Volumes are in-situ (bank) volumes (the report states this). It does not handle DWG files. It does not convert units (12.3).

12.2 Adding the node and opening the module

Step	What to do
Add	In the node library choose **Cut & Fill** (tag "Earthworks · DXF").
Open the module	Press **Open cut & fill module** on the node card, or double-click the node.
Several surveys	The header picker lists every Cut & Fill survey. Use it to switch, rename (pencil icon), delete (not offered for the last one) or **+ Add cut & fill survey**.
Undo and redo	Edits to shapes, footprint, formation level and sections can be undone and redone (40 steps). Loading a new DXF is not undoable.

The **node card** shows a status line, survey point count, cut, fill and net (the sum of every solved excavation), the file name, and the open button. Cut & Fill does not exchange data with other nodes (see 12.12).

Header status

Status	Meaning
NO TOPO	No DXF has been loaded.
NO LEVELS FOUND	The file was read but nothing in it carried a level.
NO GROUND MODEL	Fewer than three levelled points, so no TIN could be built.
DRAW AN EXCAVATION	Ground model exists but there is no shape or footprint.

Status	Meaning
FINISH DRAWING THE SHAPE / SHAPE OFF THE SURVEY / NEEDS A FORMATION LEVEL	The first thing blocking a result.
BALANCED	Net within 0.05 m³ of zero.
IMPORT x m³ / EXPORT x m³	The answer: net fill minus cut. Positive means material must be imported.

12.3 Importing the DXF

Drop a `.dxf` on the **Topo import (DXF)** panel or click it. Only **ASCII DXF** is accepted. Any other extension gives "Only ASCII .dxf files are supported - export DXF (not DWG) from CAD."

What the parser reads

From the ENTITIES section only:

DXF entity	Use
POINT	A survey point. If its z is non-zero (absolute value above 0.001) it is a levelled point with source "3d". A point with zero z is "unmatched" and waits for a text label.
INSERT (with ATTRIB)	A block, typically a spot-height symbol. A decimal level in an attribute wins; otherwise a non-zero insertion z; otherwise it waits for a label.
TEXT and MTEXT	Candidate level labels (MTEXT formatting codes are stripped).
LWPOLYLINE and POLYLINE	Contours. A 3D polyline with non-zero z, or a polyline with a non-zero elevation (group code 38), is a contour. A flat polyline whose layer name carries a level (for example a layer called like `EG-45.500`) takes that level. Any other flat polyline is kept only as drawing linework.
LINE, CIRCLE, ARC, SOLID, 3DFACE	Drawing linework only (kerbs, buildings, fences, site outline). It appears on the plan but never enters the ground model.

Priority for a point's level: block attribute, then embedded 3D z, then a nearby text label, then contour or layer-name level.

What counts as a level (text labels)

Level text must be a **standalone decimal number** with **two or three decimals** and no more than four digits before the point, inside the band -200 **m to 3000 m**. Commas are read as decimal points. Slices of longer numbers (for example the 7820.000 inside a grid annotation 537820.000), numbers with a second dot, and whole numbers are never taken as levels. A sign glued to a name (for example `EG-45.500`) is treated as a separator, not a minus.

A second, robust pass then rejects levels far from the rest of the drawing: if there are at least eight candidate levels, anything more than $\max(100\ \text{m},\ 10 \times \text{robust spread})$ from the median is dropped. The panel reports how many labels were rejected ("outside the level band") so you can see what was thrown away.

Label matching

Each point without its own level takes the level of the **nearest text label within the match distance**. A point is flagged **contested (ambiguous)** only when another label within the distance reads a *different* level (more than 0.005 m apart) and sits nearly as close (within 1.6 times the winning distance, or 0.15 m further, whichever is larger). Two labels with the same value do not conflict.

Settings in the import panel

Setting	Default	Notes
Label match distance	1.0 m	Maximum distance from a point to its level text. Changing it re-matches every text-derived point and clears the review sign-off.
Coordinate system	OSGB36 British National Grid, EPSG:27700 (assumed)	Choices are the assumed BNG or "Unknown / unverified". This is a label recorded on the survey and in the report. **Nothing is transformed**: true Easting and Northing values are used as they are, and a local origin (the minimum coordinates rounded down to the nearest 100 m) is subtracted internally for numerical stability and restored in outputs.
Contour spacing (thinning)	0.5 m	Contour polylines are thinned so that vertices are at least this far apart before entering the ground model. If the total exceeds 60,000 vertices the spacing is doubled (up to four times) until it fits. This value is not exposed as an input in the module; the report prints the value actually used.

Units

The DXF unit setting is read and stored (and counted in the file summary), but **it is not used to scale anything**. The module assumes the drawing is in **metres**. Coordinates, levels, areas and volumes are reported as m, m^2 and m^3. A survey drawn in millimetres will give wrong results. Convert to metres before export.

Other limits: the retained linework is capped (about 200,000 vertices) so very large DXFs do not overflow stored state; layer names are interned to keep the saved state small; state is stored in the browser, so a very large survey takes storage space.

Import summary

After a load the panel shows the file, entity counts (POINT, INSERT, TEXT plus MTEXT, polylines), survey points and contour vertices, levelled points against points without a level, and non-level text rejected.

12.4 DXF layers

The **DXF layers** panel lists the layers in the file with their counts of spots, labels, contours, levelled points and drawing entities. Switching a layer off removes it from the drawing **and from the ground model** (the usual "ignore the TREES layer" workflow). The TIN and all volumes are recomputed.

12.5 Point review

The ground model is built **immediately** from every point that has a level. The review is a sign-off on doubtful matches, not a gate.

Category	Meaning
Levelled	Points that feed the ground model (matched, manual or contested).
Contested	Points whose level came from a label that has a rival label. This is the only category that needs an engineer's eye.
No level (ignored)	Symbols, block nodes and drawing furniture. They are ignored automatically; nothing needs deleting.
Excluded by you	Points you removed.

For each listed point you can **type a level** (a manual override) or **exclude** it (and include it again). Manual levels beat label levels. Clicking a point on the plan selects it in the list. Bulk actions:

Button	Effect
Accept all nearest	Keeps the nearest label on every contested point and clears the flag.
Exclude all contested	Drops every contested point from the ground model.
Show / hide levelless points	Lists the points with no level so you can type levels by hand. They are ignored either way.
Reset my overrides	Clears every manual level and exclusion and returns to the drawing.

The list shows the contested points first (up to 60 rows; further points are reached from the plan).

Sign off point review records that someone has looked at the contested matches. It does not change any number. Any change to a level, exclusion or the match distance un-signs it. The report states "review signed off" or "review not signed off", and its Validation section raises a CHECK if the review is unsigned or contested points remain.

12.6 Ground model (TIN)

The levelled points (resolved spots plus thinned contour vertices, de-duplicated on a 1 mm grid, minus excluded points and switched-off layers) are triangulated by a Delaunay (Bowyer-Watson) algorithm. Inside each triangle the ground is a plane.

The triangulation fills the convex hull, so an L-shaped survey or a compound plus a road arm gets long triangles over ground nobody measured. Two controls remove them:

Control (Surface display panel)	Default	Effect
Drop triangles with a side longer than	0 (off)	Removes long triangles from the ground model entirely.
Ground-model extent (clip outline)	none	Trace the ground you actually surveyed; triangles whose centre is outside are removed.

Spike check compares each survey point with the plane through its neighbours and flags outliers (a typed decimal error such as 45.2 in a field of 4.52). Settings: Sensitivity $K = 6$ (robust, MAD-based), and "Ignore deviations below" 0.15 m. Flagged points can be excluded singly or all at once; nothing is excluded automatically. The panel warns that the check cannot tell a typo from a retaining wall or a kerb.

12.7 Surface display

Display settings never change volumes except where noted.

Setting	Default	Notes
Generate contours	on	Contours are generated from the ground model.
Minor interval	0.25 m	Raised automatically (a note says so) if too dense.
Major interval	1.0 m	Heavier, labelled on plan.
Suppress across gaps over	0 (automatic)	Contours are not drawn across long triangles.
Show in 3D / on plan / label major contours	on	
Show the imported DXF contour polylines in 3D	off	
Hillshade	azimuth $315°$, altitude $45°$	

Setting	Default	Notes
Show level values on the surface	off	Maximum 150 labels, 2 decimals.
Solid ground, cut hole, show removed ground	on	3D presentation.
Excavation shapes in 3D	on, opacity 0.75	Shapes drawn as solids. Without a build-up, a shape is coloured by cut (red), fill (blue) or mixed.

The **Delete the ground model** button (two clicks to confirm) removes the survey, triangulation and review decisions while keeping your shapes, sections and formation levels, so a corrected DXF can be dropped straight back in. The ground model can also be exported as an **IFC4 triangulated face set** (an IfcSite with the true-grid origin in its placement), so a BIM tool can place it where the survey sits.

Views: **3D surface**, **Plan & review**, and a **Split** mode with two panes side by side. Sections have their own bottom-drawer tab (**Section view**) and also appear in a stage view.

12.8 Excavation shapes, footprint and formation level

Shapes

Ready-made outlines stay editable by their parameters and by dragging corners on the plan.

Shape	Parameters (defaults)
Rectangle	width 20 m, depth in plan 10 m
Circle	radius 6 m
Polygon	radius 8 m, 6 sides
Trapezoid	top width 14 m, bottom width 8 m, length 10 m
Swale	bottom width 1.5 m, depth 0.6 m, side slope 3 (H:1V), path of two points 40 m apart
Free form	corners clicked on the plan
Pond	a stack of closed design contours picked from the DXF plan, each at its own level

Every shape has a centre (E, N), a rotation (snaps to 15° on the plan unless Shift is held), and a name. How the formation is defined:

- **By depth** (default depth 1.0 m): the formation is set that depth below existing ground **at the shape's centre**.
- **By level**: untick "Set the formation by depth below existing ground" and type a formation level in metres.
- **Side batter** (H:1V): the sides slope out from the floor; a batter "daylights" at the ground and never becomes fill.
- **Base fall**, **block thickness** and **side stand-out** let the drawn dig be shaped. Presentation-only values (block thickness) do not change cut or fill.
- A **build-up** of named layers with thicknesses can be defined (reported in the excavation definitions).

Shapes can be hidden; a hidden shape is not counted.

Building footprint (typed outline)

In the **Building footprint** panel type corner coordinates in true grid E/N (at least three), or draw on the plan. The panel shows the plan area, and warns if corners fall outside the survey extent.

Formation level

Formation level (FFL) is in metres on the survey's datum (m AOD for a UK topographic survey). For a shape with a batter, swale or fall it is the level at the shape's centre, and the formation follows the shape. A blank field is treated as absent, not zero.

IFC model

The **IFC model** panel places a building model on the survey. IFC carries no grid reference, so you enter the insertion point (E, N), rotation and base level; **Sit the base on existing ground** reads the ground at the insertion point. **Dig for every buried element** makes each element whose base is below the ground an excavation (the model's footprint can also be used as one outline, or split into separate buildings further apart than a set distance, with the convex hull of each block in plan). Overlaps are separated approximately; **Work harder on the overlaps** raises the solver effort. A row flagged "overlaps not fully separated" is over-stated.

12.9 Volume method

Volumes use the **prismatic TIN method**:

1. Each excavation outline is clipped against every ground triangle it touches.
2. Each clipped piece is split into triangles whose three corners have a height difference $h = \text{formation} - \text{ground}$.
3. The volume of each prism is $V = A \times (h_1 + h_2 + h_3)/3$, with A the triangle plan area.
4. Triangles that cross the formation (some h positive, some negative) are split along the zero line into a pure-cut and a pure-fill part.
5. Cut is the sum where the ground is above the formation, fill where it is below.

$$\text{net} = \text{fill} - \text{cut}$$

A positive net means import, negative means export. This is exact for a planar ground triangle and a planar (or, for shaped formations, per-corner-height) formation. The report quotes the method as the Prismatic TIN volume method and compares it to the Civil 3D TIN volume surface approach.

The module also reports:

Result	Meaning
Cut / Fill (m^3) with plan areas	Plan area over which the ground is above or below the formation.
Max cut depth / Max fill depth (m)	Deepest prism corner.
Ground model coverage (%)	Footprint area covered by ground triangles. Below 99.5% the module warns that volumes outside survey coverage are not counted.
Extent check	Footprint corners outside the survey extent give a FAIL ("extrapolated volumes are unreliable").
Mean cut / fill depth	Volume divided by plan area.

Whole-site total

Every solved shape is solved independently against the ground model and the results are summed (the header chip and card show the site total, and say "n excavations summed"). Shapes that could not be solved (not drawn, off the survey, no formation) are excluded from the sum and counted as unsolved. If no shape solves, the typed footprint with its formation level is used. **Overlapping shapes are each counted in full**, so the total is a true sum only where they do not overlap (the report notes this).

Sections

Draw a section line on the plan. The profile is cut from the TIN with a breakpoint at every triangle edge crossed, and gives a ground line, the shapes it crosses, and **cut and fill end-areas** in the vertical plane (m^2). End-areas are not plan areas and must not be used to derive a volume.

12.10 Outputs

Output	Where	Unit
Cut, Fill, Net	Node card, header chips, Cut & fill results panel, right panel	m^3
Excavation area	Results, report	m^2
Max cut and max fill depth, mean depths	Results panel and report calculation table	m
Coverage	Results panel, report	%
Per-excavation take-off	Right panel and report	m^3 each
Section end-areas	Sections panel, report	m^2
Contours	Plan and 3D	at the set intervals
IFC export of the ground	Module	IFC4 triangulated face set

12.11 Report

Open the module and use the header **Report** button. It includes:

- Headline (cut, fill, net and number of excavations), metric cards and a summary table (cut, fill, net, excavation area, excavation count).
- *Topo survey and ground model*: source file, points, levels used, rejected annotation, contour vertices and thinning, TIN size and sign-off state, local origin, CRS, layers, contour interval, spike check settings and any placed IFC model.
- *Building footprint* corners in true grid coordinates, *Cut and fill volumes*, the per-excavation table with a TOTAL, and *Excavation definitions* (kind, depth or level, batter, fall, swale geometry, build-up).
- *Calculation, selected excavation*: plan area, formation level, tiles integrated, the cut and fill formula, net, mean and maximum depths, and coverage.
- *Sections* with end-areas, each drawn as a figure.
- *Validation* rows (OK, CHECK, FAIL): no levels found, points with no level (ignored), rejected labels, contested points, review not signed off, unsolved shapes, extent failure and low coverage.
- Figures: the plan (survey, contours, excavations and section lines), a 3D capture if the 3D view is open, and one figure per section.
- References printed: the prismatic TIN method, BS 6031:2009 (earthworks), BS 5930:2015+A1:2020 (ground investigations), and OSGB36/British National Grid (EPSG:27700), the last as assumed.

The report is the deliverable for the whole site. It prints the summed site total, not just the selected shape.

12.12 Connections to other nodes

Cut & Fill is a stand-alone module. It has no wire ports, so nothing flows in or out through wires. It has no carbon contribution. The only inter-module exchange is the IFC export of the ground model (a file) and the IFC model panel, which loads an IFC file directly into this module (it is not linked to an IFC viewer node). Cut & Fill nodes appear in the project-wide report list in the order prelim, wind, slab, beam, column, baseplate, foundation, cut & fill, IFC.

12.13 Tips and limitations

- Export **ASCII DXF in metres**, with levels as 2 or 3 decimals. Spot heights as block attributes or 3D points are the most reliable. Level text is the least reliable source: check the contested list.
- If many points show "no level", raise the label match distance slightly (and re-check contested points).
- Use the spike check after import, then switch off layers such as trees or services that are not ground.
- Put the formation level on the survey datum. A blank formation means no result, not zero.
- Overlapping shapes double count. Draw one outline per separate dig, or use one free-form shape.
- Volumes are bank volumes; apply bulking and compaction yourself.

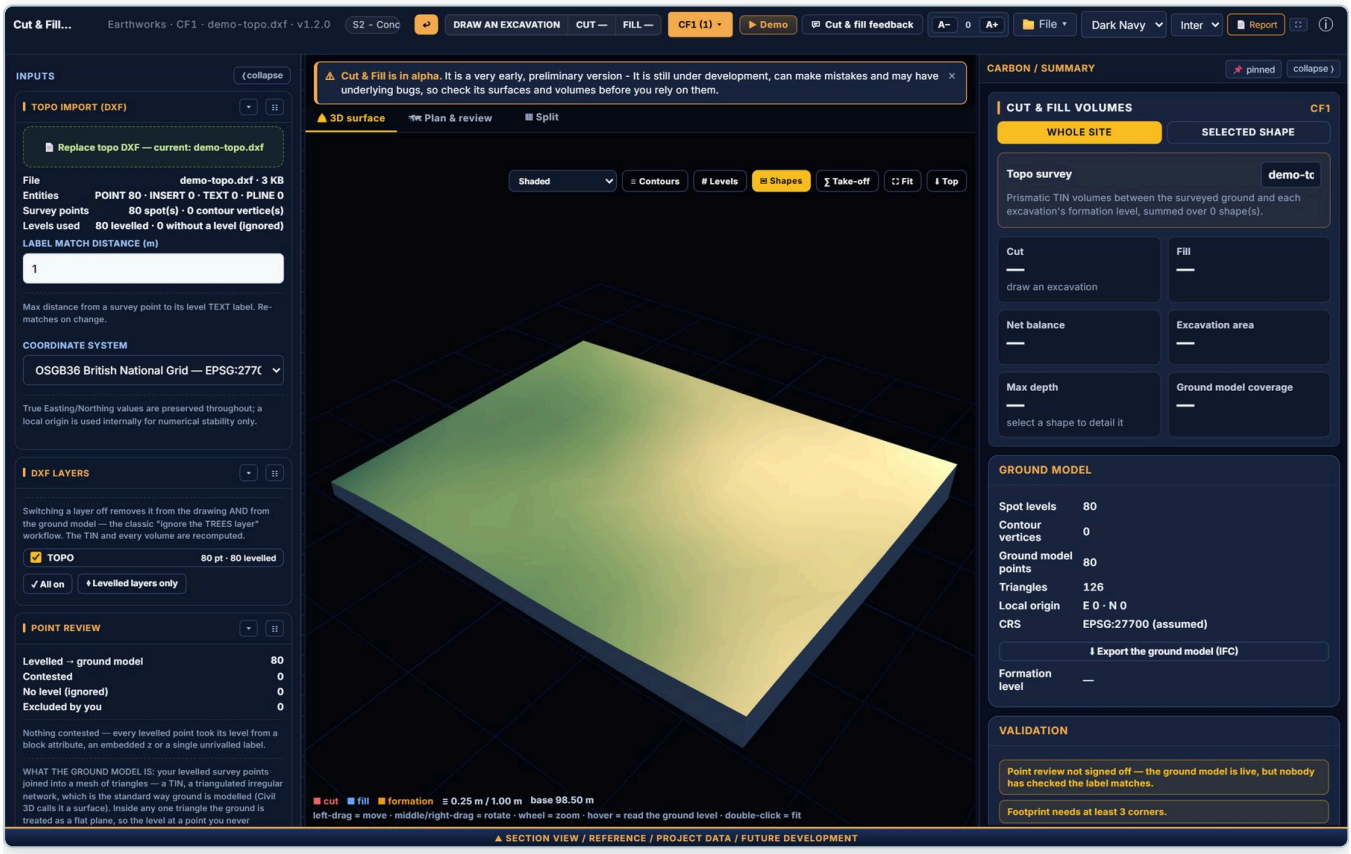
- The CRS selector is a label. No coordinate transformation is done.
- Survey points can only be used if they fall inside the ground model. Keep the footprint inside the survey extent (the module warns otherwise).
- Alpha status: expect behaviour to change between versions.

> **References.** The standards, guidance and data sources used in this chapter are listed, with web links, in Chapter 17.

Screen reference - panel by panel

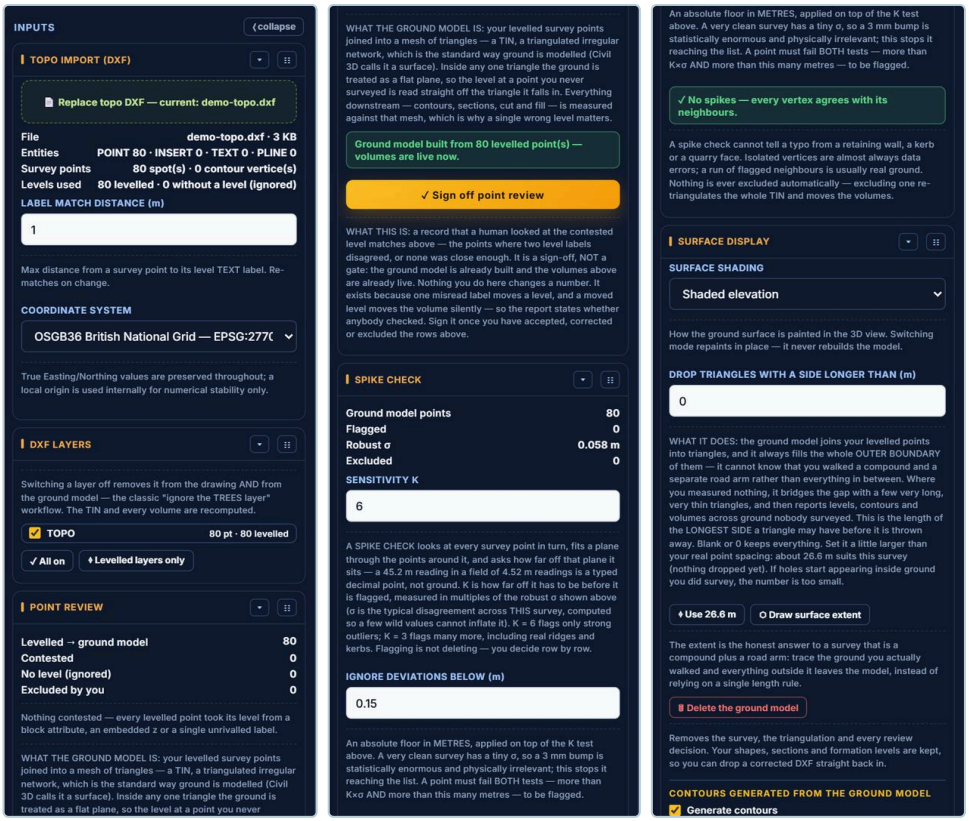
These screenshots show every panel of the module exactly as it appears on screen, scrolled from top to bottom, so you can match any field, button or result to the explanations in this chapter. Cards that are collapsed by default are shown expanded.

Cut & Fill with a sample survey: the whole screen

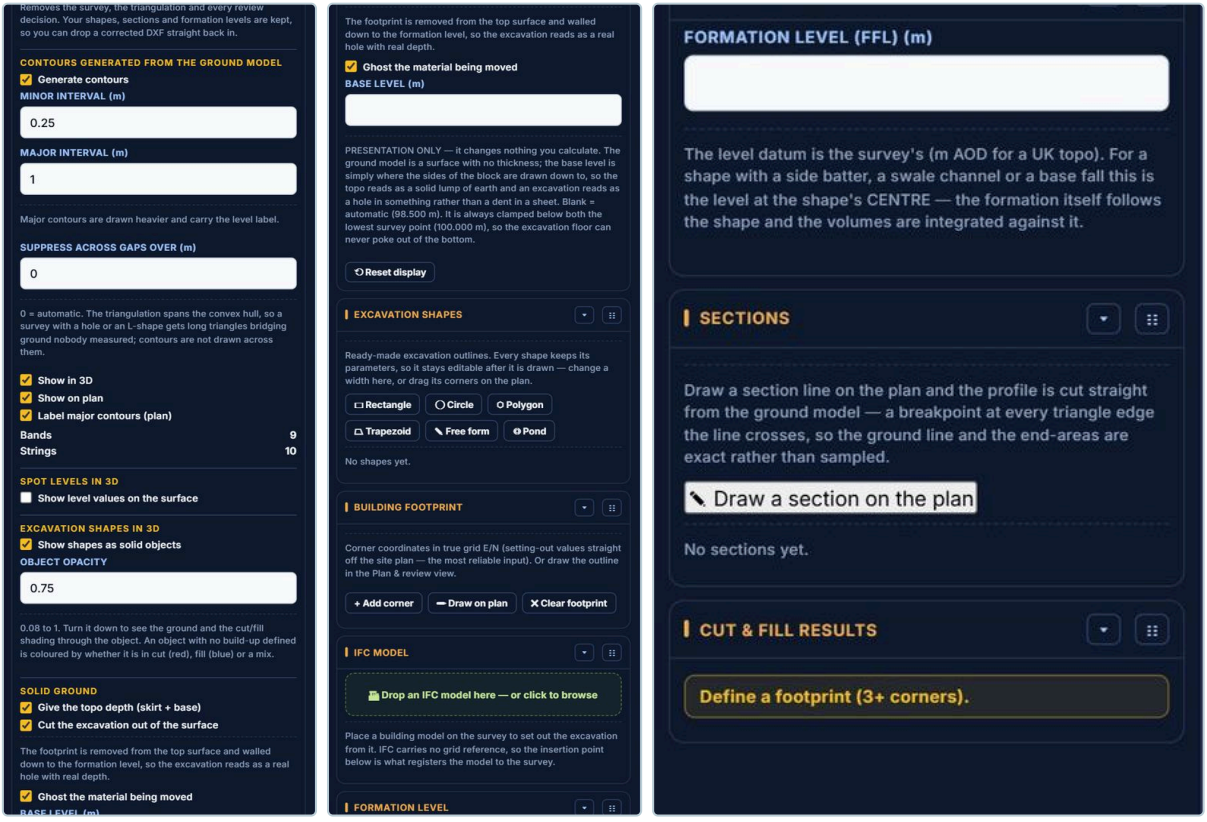


Cut & Fill with a sample survey: the whole screen.

Cut & Fill with a sample survey: input panel (left of the screen)

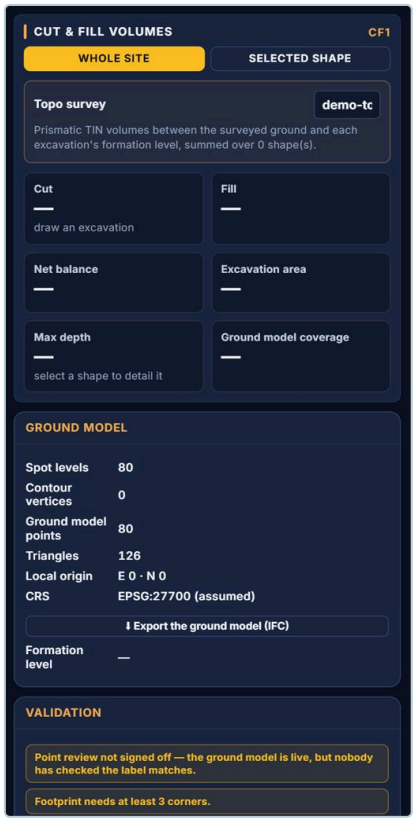


Cut & Fill with a sample survey, input panel (left of the screen), parts 1-3 of 6. Cards shown: Topo import (DXF), DXF layers, Point review, Spike check, Surface display.



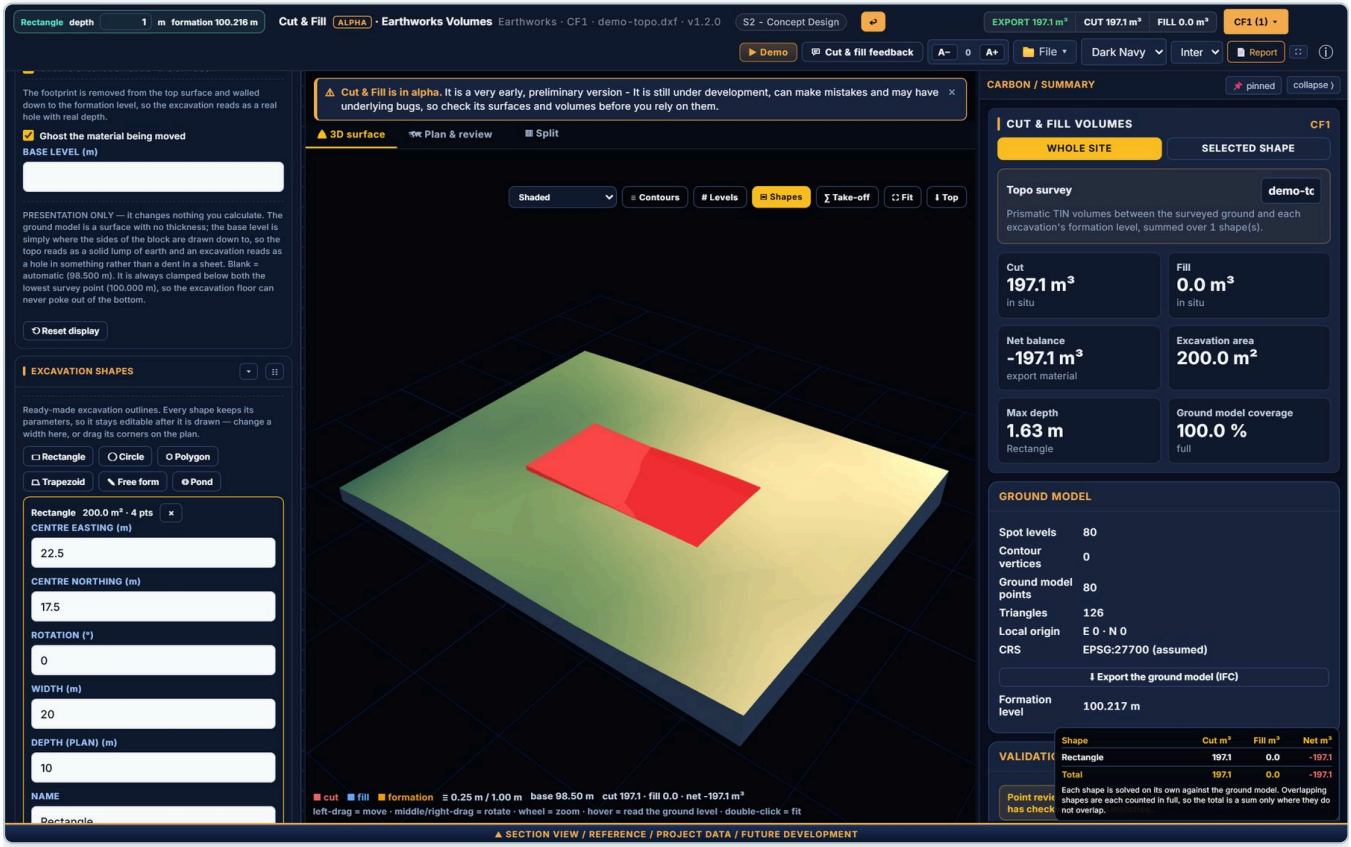
Cut & Fill with a sample survey, input panel (left of the screen), parts 4-6 of 6. Cards shown: Excavation shapes, Building footprint, IFC model, Formation level, Sections, Cut & fill results.

Cut & Fill with a sample survey: results panel (right of the screen)



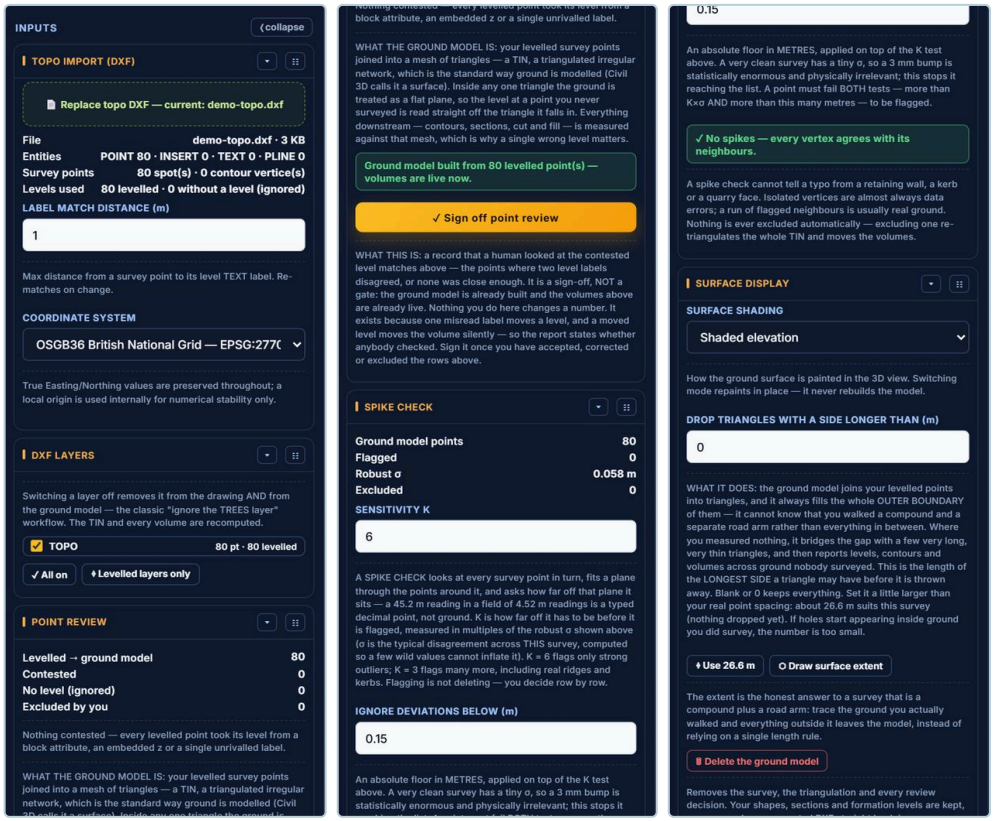
Cut & Fill with a sample survey, results panel (right of the screen), part 1 of 1. Cards shown: Cut & fill volumes CF1, Ground model, Validation.

Cut & Fill with an excavation shape: the whole screen

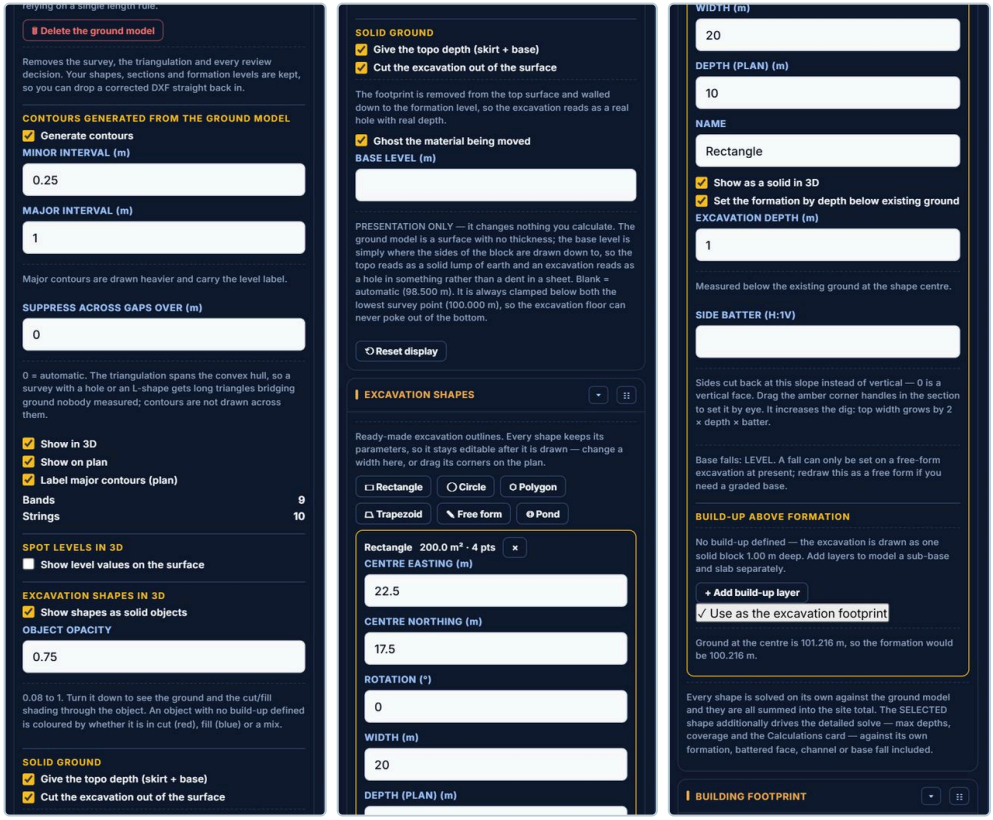


Cut & Fill with an excavation shape: the whole screen.

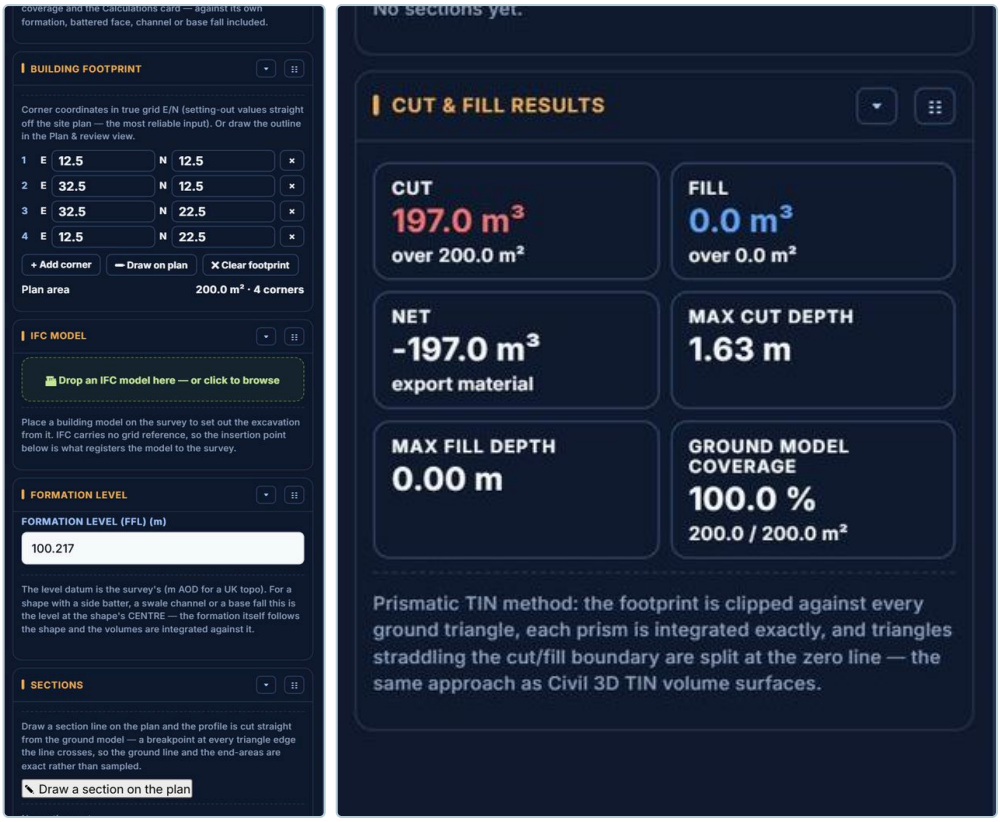
Cut & Fill with an excavation shape: input panel (left of the screen)



Cut & Fill with an excavation shape, input panel (left of the screen), parts 1-3 of 8. Cards shown: Topo import (DXF), DXF layers, Point review, Spike check, Surface display.

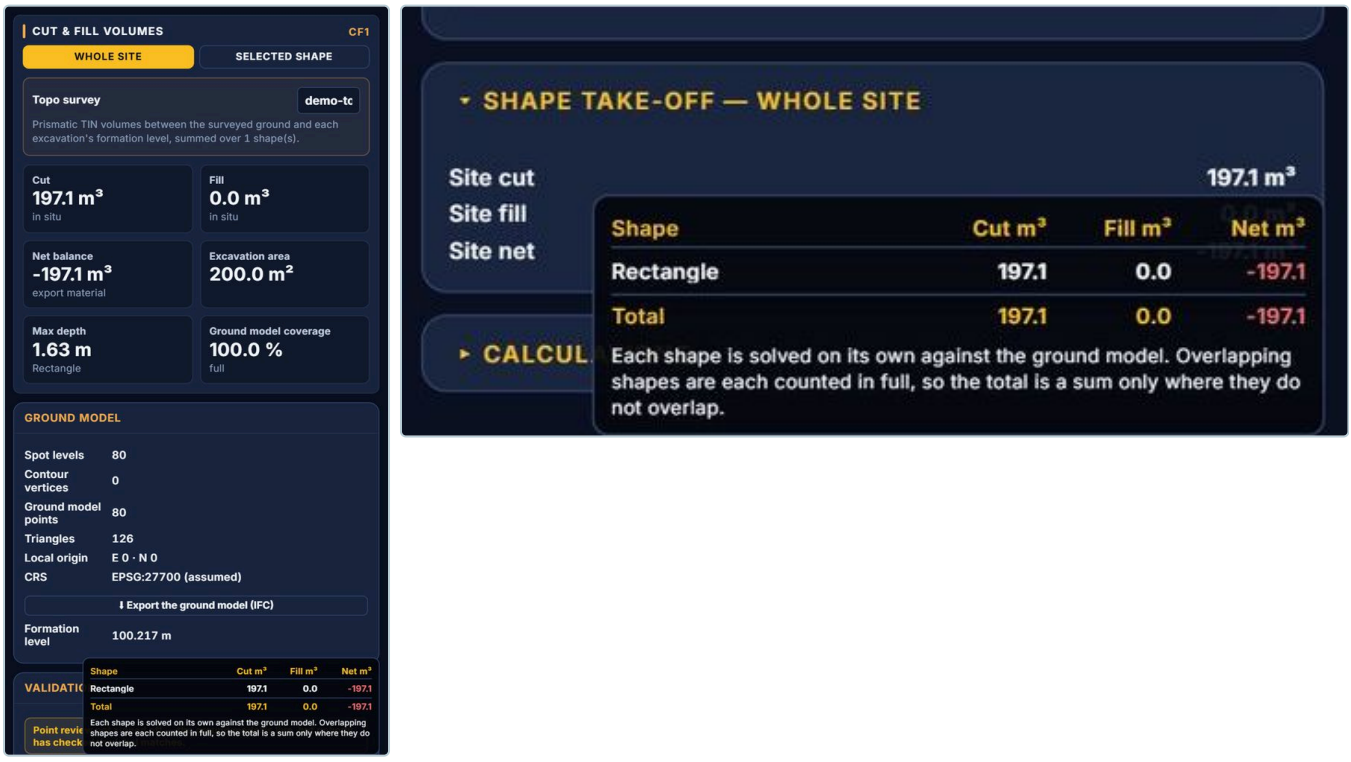


Cut & Fill with an excavation shape, input panel (left of the screen), parts 4-6 of 8. Cards shown: Excavation shapes, Building footprint.



Cut & Fill with an excavation shape, input panel (left of the screen), parts 7-8 of 8. Cards shown: Building footprint, IFC model, Formation level, Sections, Cut & fill results.

Cut & Fill with an excavation shape: results panel (right of the screen)



Cut & Fill with an excavation shape, results panel (right of the screen), parts 1-2 of 2. Cards shown: Cut & fill volumes CF1, Ground model, Validation, Shape take-off — whole site.

CHAPTER 13

SI map

The SI map node puts site investigation (SI) reports on a map. Drop in your company's PDF or AGS reports and Civil ET reads out the borehole locations, the soil at the founding depth and the bearing values the report quotes, so you can see what is known about the ground around a site before design starts. The module is marked **ALPHA** in the app and is a desk-study aid, nothing more.

> **Indicative only, not for design.** The SI map shows values as they were written in the source documents. Civil ET does not model, interpolate into or predict ground behaviour, and the module says so on every screen and in the summary panel. Always take design bearing values from the project's own ground investigation and the geotechnical engineer. Nothing on the SI map feeds the Foundation node: type the bearing value you decide to use into the foundation design yourself.

13.1 The SI map card

Add **SI map** from the node library (tag *Ground · SBC*). The card shows:

- the workspace name (your company library, called *My company* until you rename it);
- how many reports are on the map and how many have been reviewed;
- the number of boreholes and of bearing values;
- whether BGS records are available (they are shown when you zoom in);
- when the library was last reviewed;
- the permanent badge *Indicative only · not for design*, and a notes box.

Open SI map opens the full module. Like every node card it has the guided-tour button, the feedback button, the sync button, rename, minimise and delete.

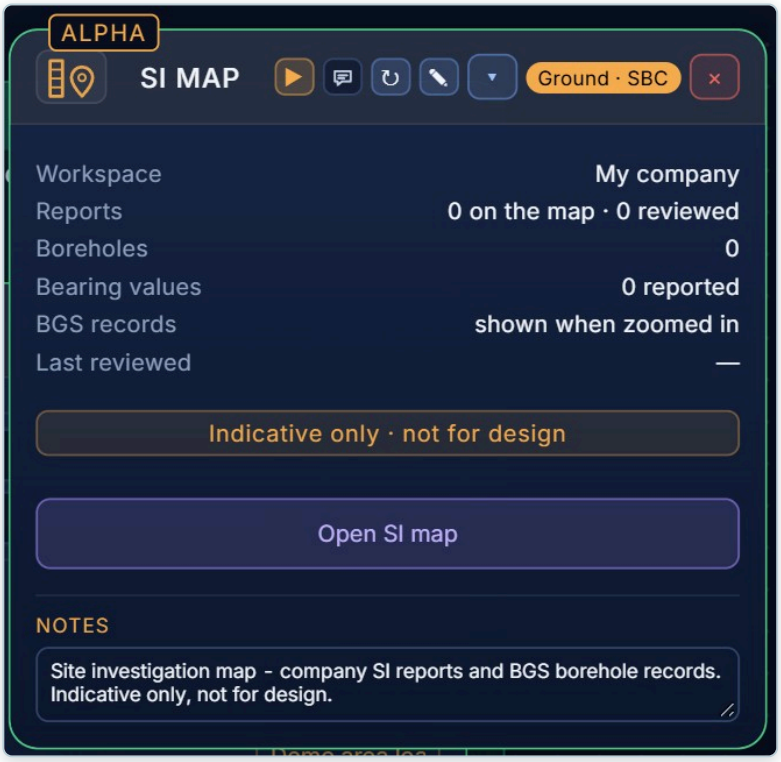


Figure 13.1 The SI map node card: library totals, the permanent "Indicative only" badge and a notes box.

13.2 Layout of the module

The module follows the same four-panel layout as every other module (Section 1.5). In the header, live counters show **REPORTS** (on the map over the total), **HOLES** and **SBC** (bearing values found). The middle panel has two tabs, **Map** and **Review**. The left column holds the cards you work with: Add SI reports, Reports, Map layers, Filters, Nearby boreholes, Heat layer and Library & privacy. The right column summarises the library.

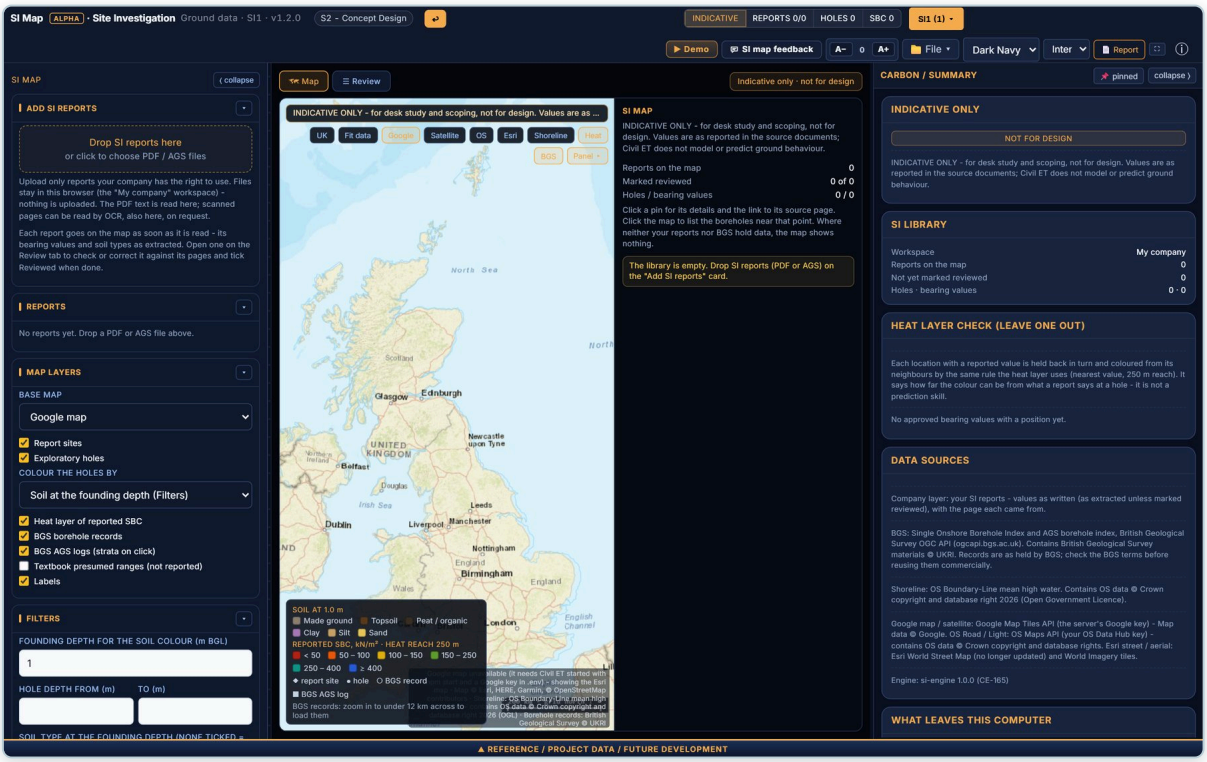


Figure 13.2 A new, empty SI map: the Add SI reports drop zone, map layers and filters on the left, a map of the UK in the middle and the library summary on the right.

13.3 Adding reports

Drop one or more **PDF** or **AGS** files on the *Add SI reports* card, or click it to choose files. Each file is read as soon as it is added:

- **PDF reports** are read for the site address or grid reference, the borehole and trial-pit references, the soil at depth and any quoted safe, allowable or presumed bearing values. Scanned pages can be read by OCR when you ask for it.
- **AGS files** (the standard geotechnical data interchange format) give exploratory-hole locations and strata directly.
- The report goes on the map immediately *as extracted*; it is then up to you to review it (Section 13.5).

A progress row for each file shows how many holes and bearing values were found and how many flags need your attention. **Clear finished** tidies the list away once files are done.

Upload only reports your company has the right to use. Files stay in your browser, in a workspace called *My company*; nothing is uploaded to a server.

13.4 The map

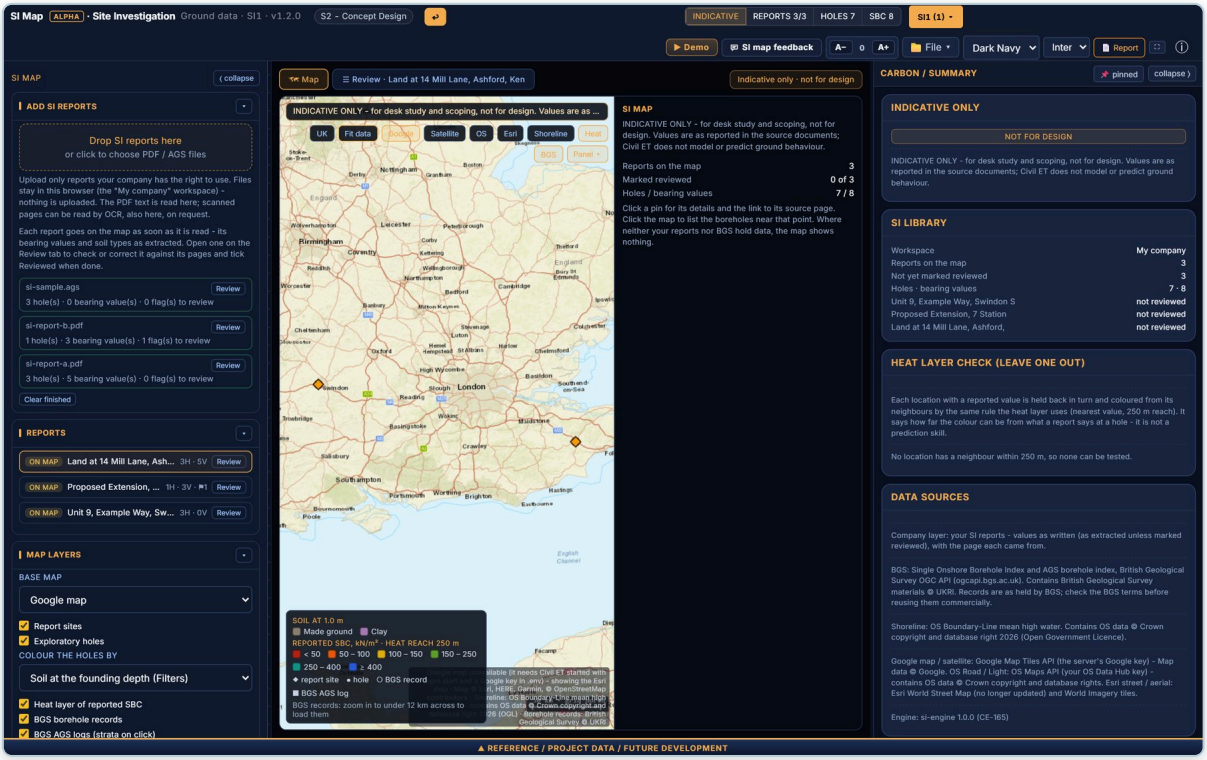


Figure 13.3 Three sample reports loaded: report sites appear as pins, exploratory holes are coloured by soil at the founding depth and the legend at the bottom left of the map explains the colours. The header shows 3 reports, 7 holes and 8 bearing values.

The buttons across the top of the map jump around: **UK** shows the whole country, **Fit data** zooms to your reports, and **Google**, **Satellite**, **OS**, **Esri** and **Shoreline** switch the base map. **Heat** and **BGS** switch the heat layer and the BGS records on and off, and **Panel** hides the side panel. Click a pin for its details and a link to the page of the source report it came from. Click anywhere on the map to list the boreholes near that point.

Map layers

Layer	What it shows
Base map	Google map or satellite (where a Google map key is connected; otherwise the Esri map is shown instead), Ordnance Survey Road and Light, an Esri street map, Esri aerial imagery with an OS shoreline, or an offline OS shoreline only.
Report sites	One pin per report at the site location.
Exploratory holes	The boreholes and trial pits listed in your reports, coloured by the option below.
Colour the holes by	**Soil at the founding depth** (using the filters), **Lowest reported SBC**, or **Heat layer of reported SBC**.
BGS borehole records	Public borehole records from the British Geological Survey, shown when you are zoomed in to within about 12 km.
BGS AGS logs	Where BGS holds an AGS log for a hole, click it to see the strata.
Textbook presumed ranges	Optional generic ranges, clearly labelled as not reported. Off by default.
Labels	Hole references and values on the map.

Filters

The *Filters* card controls what the soil colouring and the legend show:

- the **founding depth** (m below ground level) at which the soil is read;
- a **hole-depth range** (from and to, in m);
- the **soil types** to include: made ground, topsoil, peat or organic, clay, silt, sand, gravel, cobbles and boulders, chalk, rock, other or unclear (none ticked means all);
- a **reported SBC range** (from and to, in kN/m²);
- the **bearing value basis** to show: allowable or safe, presumed, ultimate, design (EC7) or basis not stated.

Allowable and ultimate values are never mixed on one colour scale unless you tick both. With no basis ticked no value is shown. A value keeps its footing width and depth beside it wherever it is shown.

Nearby boreholes

Click the map, or type a reference and press **Go**, or use **Look up postcode**, to list the boreholes within a chosen **search radius** (m). The result can be exported as a **CSV desk-study table** or as a **Desk-study report (PDF)**.

13.5 Reviewing a report

Press **Review** beside a report in the Reports list. The Review tab shows every extracted value next to the page it came from.

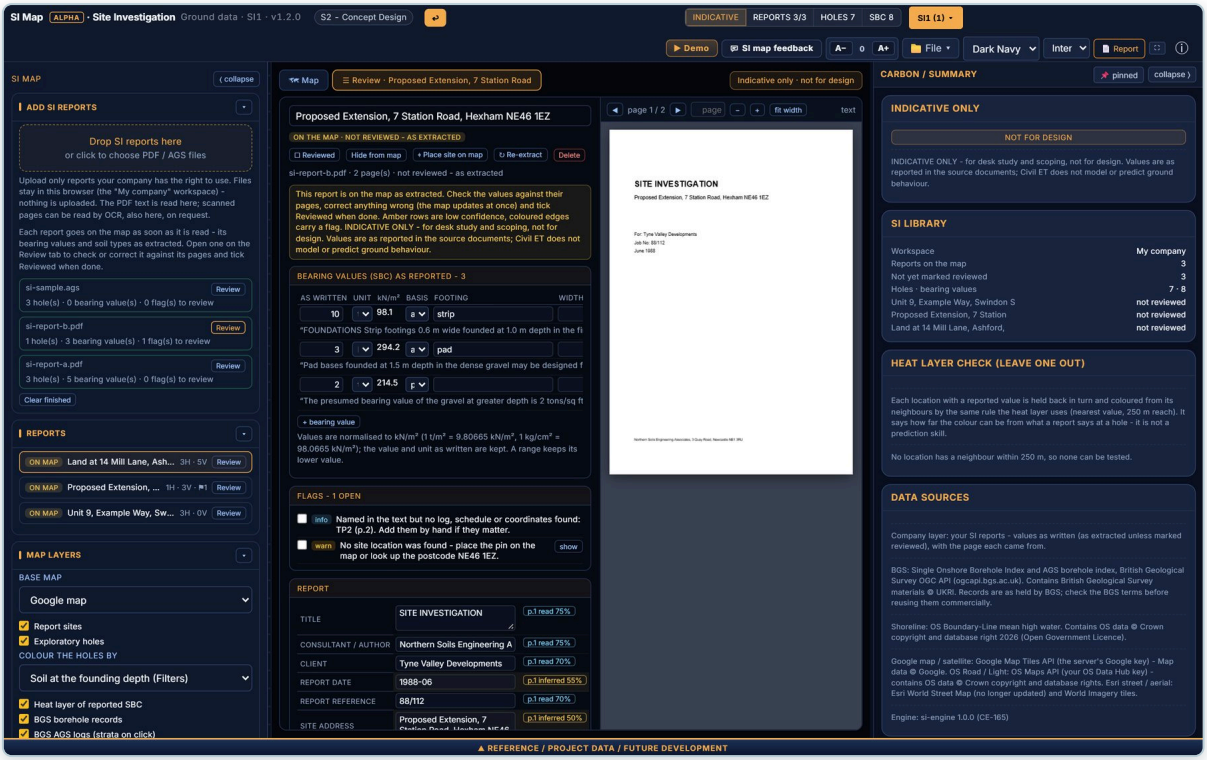


Figure 13.4 The Review tab: the extracted values on the left; the source page, with page controls and fit width, on the right.

- **Status and actions.** Each report is marked *On the map · not reviewed · as extracted*. **Reviewed** marks it as checked, **Hide from map** removes it from the map without deleting it, **+ Place site on map** lets you click the location when none was found, **Re-extract** reads the file again and **Delete** removes it.
- **Bearing values (SBC) as reported.** Each row keeps the value *as written* and its unit, and shows it normalised to kN/m², with the basis, the footing type and size and the sentence quoted from the report. Values are normalised using 1 t/m² = 9.80665 kN/m² and 1 kg/cm² = 98.0665 kN/m². A range keeps the lower value. **+ bearing value** adds one by hand.

- **Flags.** Anything the reader was unsure of is listed with a severity (*info* or *warn*) and a **show** button that jumps to the place. Amber rows are low-confidence reads.
- **Report details.** Title, consultant, client, date, reference and site address, each with where it was read from and how confident the read was (for example *p.1 read 75%* or *p.1 inferred 55%*). Correct anything that is wrong and the map updates at once.

13.6 The summary panel

The right-hand panel replaces the carbon summary with ground-specific cards:

- **Indicative only**: the permanent *Not for design* reminder.
- **SI library**: the workspace name, reports on the map, reports not yet reviewed, holes and bearing values, and each report's review status.
- **Heat layer check (leave one out)**: see the next section.
- **Data sources** and **What leaves this computer**: the attributions for the data shown and a plain statement of what is sent over the network.

13.7 The heat layer and its check

The **heat layer** colours the map by the bearing values your reports state, and only where they state them. There is no interpolation and no colour where there is no data. Two settings control it:

- **Reach**: nothing is coloured further than this distance (m) from a reported value.
- **Colour by**: *the nearest reported value*, or *the lowest reported value within reach* (the conservative choice). You can also limit it to values that state a founding depth.

Hover the map to see how many values sit behind a spot. The **leave-one-out check** in the right-hand panel holds each location back in turn, colours it from its neighbours by the same rule, and reports how far the colour can be from what the report itself says. It tells you how far to trust the colour; it is not a prediction.

13.8 Library and privacy

The **Library & privacy** card holds the company workspace name and the totals: reports, those on the map, those reviewed, and approved holes and bearing values. **Export library (.json)** and **Import library** let you keep a copy or move it to another computer.

The library lives only in the browser profile you are using. It is not part of project files, and clearing the browser's site data deletes it, so export it to keep a copy.

Reports, pages, values and files are never sent anywhere. What does leave the computer is the map extent you are looking at (to the British Geological Survey for the public borehole records, and to the map tile provider) and a postcode when you press **Look up postcode**. OCR, when you ask for it, downloads the Tesseract engine from a public content network and runs on your computer.

13.9 Tips and limitations

- Review every report before relying on a value: extraction from PDF text can misread a unit, a footing size or a depth, and flags only catch some of these.
- Bearing values quoted in different reports may refer to different footing widths, depths and bases. Compare like with like, using the filters.
- The map shows nothing where neither your reports nor BGS hold data. That means "no data", not "good ground".
- A report with no site location stays off the map until you place it (**+ Place site on map**) or the postcode lookup finds one.

CHAPTER 14

Reports

14.1 Overview

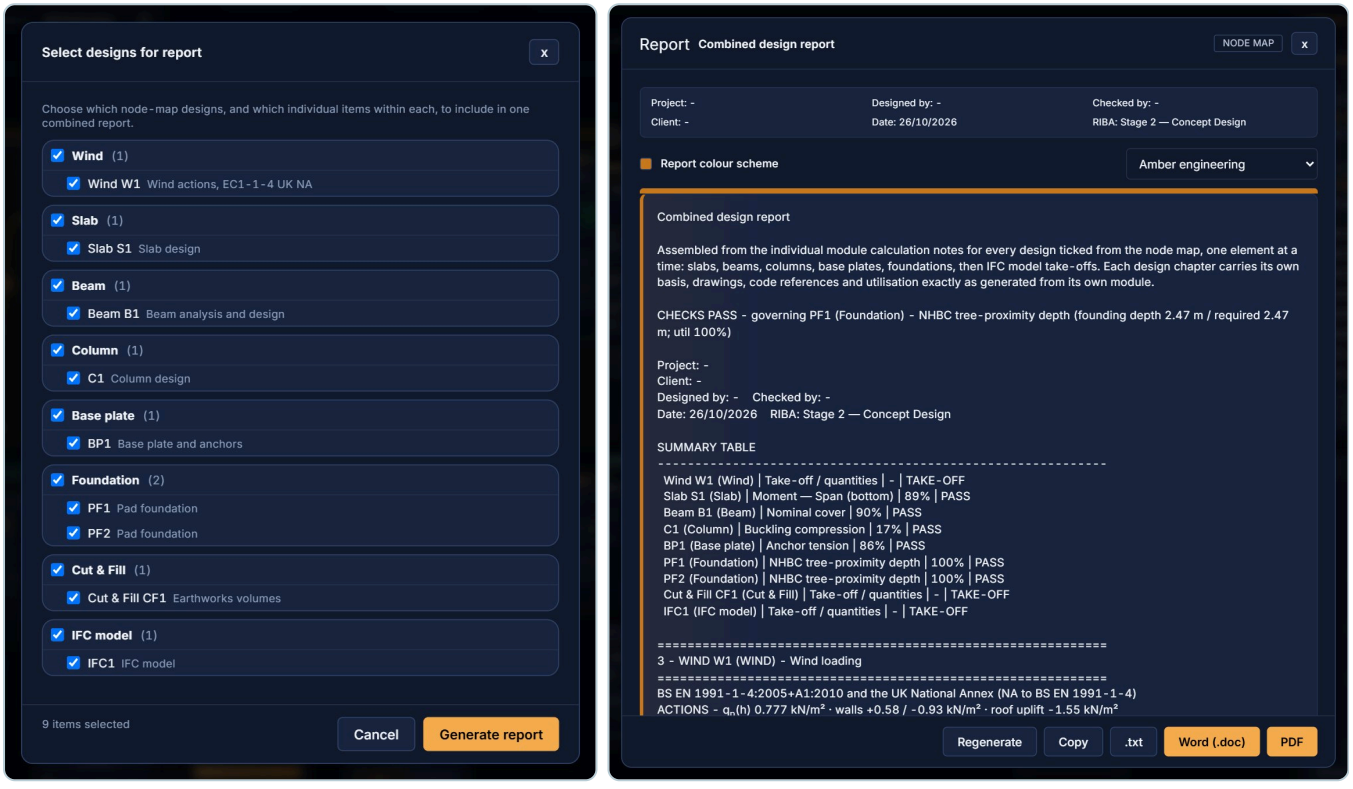
Every design module in Civil ET produces a calculation note from the live design. All of them use one house style, set by the foundation designer's detailed report: a cover with a coloured pass/fail block, numbered chapters that each start on a new page, working tables with symbol, working and result columns, figures with captions, and a closing disclaimer.

The note can be previewed on screen, copied, saved as text, saved as a Word document (`.doc`), or saved as a PDF through the browser's print dialogue. A combined report can gather several designs from the node map into one document.

This chapter explains how to produce each report, what every report contains, which modules produce what, and the rules that keep reports honest. See the module chapters for the checks behind each number.

14.2 Producing a report

The Report button



Screenshot: The node map Report button: select which designs to include (left), then the combined design report with its export buttons (right).

The header carries one **Report** button. What it does depends on where you are.

You are in	Report opens
The node map	A picker, "Select designs for report", listing every design on the map; the result is one combined report (see below)
Foundation designer	The foundation report window (its own, richer window; see below)
Column, Base plate, Beam, Slab, IFC viewer, Cut & Fill, Wind designer or Sketch-ET	That module's report window

If the module's report cannot be generated, an alert says so and asks you to check the module inputs. In the foundation designer the button is disabled while a calculation is pending or has failed.

The report window

The window shows the project block (project, number, designer, checker, client, date, RIBA stage), a colour scheme drop-down, an optional tick-list of report options (the wind report offers its full-working appendix this way), a text preview and, under it, the figures that go into the Word file ("Views and design diagrams").

Control	Effect
Report colour scheme	Amber engineering (default), Black / white, Corporate blue, Reading comfort. The scheme tints headings, table heads and the cover rule in the preview and in the exported file
Regenerate	Re-reads the live design and redraws
Copy	Redraws, then copies the text note to the clipboard
.txt	Redraws, then downloads the plain-text note
Word (.doc)	Redraws, then downloads the Word document (a Word-compatible document, which Word opens with the page set-up and page breaks)
PDF	Redraws, then opens the print dialogue with the finished report; choose Save as PDF

The foundation window also offers a **Report mode** (Detailed or Simple report) and a free-text **Calc description / scope** that appears in the report header.

Project details come from "Project info & metadata" in the file menu. Fill them in once and every report carries them.

Live data

A report is never a stored copy. The window redraws the note each time you press Regenerate, Copy, .txt, Word or PDF, so what leaves the app is the design as it is now. If you edit the design while the window is open, it redraws itself about a third of a second after the last change. Options ticked in the window are passed to the refresh, so the report rebuilds with or without them.

The combined report is live in the same way: each draw or download re-reads every chosen design and recomputes the carbon totals.

Word, text and PDF

- **Word** (`.doc`): the document Civil ET writes. Figures are embedded as PNG images (Word cannot embed SVG) with explicit width and height. Page margins and a thin page border are set in the document.
- **PDF**: the same document is printed by the browser, so chapters, figures and page breaks are identical and the text is selectable. The suggested file name is the report's title. If the browser refuses to print a frame, the document opens in a new tab instead; if pop-ups are blocked you are told to allow them.
- **Text** (`.txt`): a plain-text rendering of the same data for pasting into other documents.

Every number in the exported file passes through the same finishing step (see "House style"), whichever format you choose.

14.3 House style

All module reports share one look. The foundation designer has its own report window with the same look.

Cover and status block

The cover shows the module's title in capitals, "subtitle - Civil ET v1.2.0", the standard line (the codes the note works to) and a project box (project number, name and RIBA stage). Beneath it is a coloured block:

- green tick: **CHECKS PASS**

- red cross: **CHECKS FAIL - REVIEW REQUIRED**, followed by the governing check and its utilisation
- blue "i": take-off reports with no pass/fail (the IFC quantities report), which print "TAKE-OFF" and a headline instead

Chapters

Chapters are numbered "N - Title", and each starts on a new page except the first. The common set, in order:

Chapter	Contents
Project Details	Project card and a table of project, number, client, designer, checker, date, RIBA stage, element and module
Summary of Checks (or Summary of Designs, in a combined report)	Table of description, design action, resistance, utilisation and a PASS / FAIL / REVIEW pill; optional metric cards
Carbon Rating	IStructE SCORS rating figure (modules that carry it)
Quantity and Carbon Split	Donut charts for take-off reports
Views and Design Diagrams	The figures, numbered and captioned "Figure n - title"
Working Calculations	Sub-sections numbered "N.m - title"
Appendix - Full Working	Optional
Sustainability and Embodied Carbon Assessment	A1 to A3, A4, A5 table and charts
References and Design Basis	Reference, clause or source, and use in the calculation

A module may ask for its drawings after the working or add an appendix chapter. The pill colours are green for PASS, red for FAIL, amber for REVIEW (an item flagged for the engineer but not failed, for example a cover below the BS 8500 durability minimum).

Disclaimers

Every report ends with two statements, taken from the foundation module so they cannot drift: the responsibility statement ("Report generated by Civil ET analysis software. All calculations and design decisions remain the responsibility of the engineer ...") and the embodied carbon statement (indicative only, based on industry averages). The footer repeats the first with the generation date, the version and the module name.

14.4 Gating - a report prints only what the design contains

The user rule is that a report follows the actual design: every printed number comes from the user's own design, and nothing is printed from an input fallback or a stale value. The mechanisms:

Rule	Where it applies
Result flags decide content. Reinforcement content is printed only if the design produced it: the reinforcement-design option for pads, strips and combined footings, analysis-only mode for pile caps. A pad with reinforcement design off is reported as plain concrete designed to EC2 12.9.3 and prints no bars, flexural steel, crack check or bar schedule; an unreinforced strip likewise; a pile cap in analysis-only mode reports pile reactions and EC7 pile checks and states that the EC2 cap design was not carried out	Foundation
Excluded modules are named. A calculation module switched off in the calc options (bearing, sliding, uplift, punching, settlement, NHBC) is listed in the scope as excluded. Reinforcement is not listed there; what was designed in its place is stated once	Foundation
Pass/fail is computed. The status block uses the module's own verdict where it gives one, otherwise it fails if any summary row ends in a failure and the governing utilisation exceeds 1. A combined report passes only if every included design passes	All
Mode is named. The foundation title and cover say "Detailed report" or "Simple report"	Foundation

Rule	Where it applies
No report mid-calculation. The foundation Report button is disabled, and an open window closes, while the result is pending or failed. A node-map foundation chapter uses only a settled result; if there is none it asks the foundation calculation the same way the designer does	Foundation, combined
Take-off reports have no verdict. The IFC quantities report carries no verdict, so no "CHECKS PASS util 0%" banner appears	IFC
Rounding is for display only. Calculations use full precision; the 3 decimal rule applies to the printed text	All

14.5 What each module's report contains

All module reports share the cover, project chapter, summary table, views, working, carbon and references chapters described above. The specific content follows.

Foundation

Produced by the foundation designer's own report window, because it needs its live views for its figures. Titles follow the type: Pad foundation, Combined footing, Strip footing, Pile cap, with "Analysis and design", EC7 DA1 / EC2, and the Eurocode edition lines on the cover.

Detailed report chapters: Project Details; Summary of Checks (with the foundation size); Foundation Views and Design Diagrams; Input Data; Working Calculations (every check with substituted numbers, plus the bar schedule for reinforced designs and the pile reaction schedule for pile caps); Sustainability and Embodied Carbon Assessment; References and Design Basis.

Simple report chapters: Project Details; Summary of Checks; Foundation Views; Key Inputs; Carbon Summary; References and Design Basis.

Figures (detailed): 3D view (the live viewer if it is showing this foundation, otherwise a drawn 3D view); B/W detail X-X with plan, and Y-Y; the plan with base pressure; sections X-X and Y-Y; design shear-force and bending-moment diagrams for each axis (where bars are designed; for a strip across the width per metre run); the NHBC graphic if the NHBC module is on. The simple report drops the Y-Y detail and the sections.

A text version of the note is also produced (Copy and .txt), with the same sections as plain text.

Column

Calculation note for the steel or reinforced concrete column. Chapters include design actions, section properties and classification, cross-section and member buckling resistance for steel (EN 1993-1-1), and axial resistance, bending about each axis including second-order effects (5.8), biaxial bending (5.8.9), shear (6.2), detailing (9.5) and fire resistance (EN 1992-1-2 Table 5.2a) for concrete, with the optimised column options when found. Figures: section, elevation and design actions, N-M interaction diagram, 3D view, and the carbon charts.

Base plate

EN 1993-1-8 component method: design forces, column / base plate / foundation data, T-stub forces, compression and tension T-stubs, bolt group, moment resistance (Table 6.7), welds (4.5.3), anchors (ETAG 001 / TR029), shear transfer, quantity and carbon. Figures: the plate, section and actions as drawn on screen, and carbon charts.

Beam

Beam analysis note: model, supports, loads, support reactions, peak envelope, actions and combinations (EN 1990), analysis working, then either the EC2 concrete beam design (anchorage and lap lengths, durability cover, curtailment and end anchorage) or the EC3 steel beam design (section classification, lateral-torsional buckling per restraint segment, web bearing and buckling, deflection to the UK NA for each span). Figures: loads, shear force, bending moment and deflected shape, and the proposed concrete or steel cross-section.

Slab

Slab model, warnings and notes, working, optimised sections, quantity and carbon. Figures: plan with supports and bar layout to scale, Section A-A, design diagrams (bending moment, shear force, deflection, required reinforcement), 3D view and the reinforcement heat maps for the bottom and top faces.

Wind

Wind actions to BS EN 1991-1-4 and the UK National Annex: design inputs, peak velocity pressure by direction, checks and assumptions, force coefficients, a worked calculation per direction, worst net pressure for cladding and fixings, the structural factor, funnelling checks on the side walls, and flow visualisation. Figures include the site on the UK wind map and the fetch to shoreline, pressure zones and the 3D view as on screen. The wind node offers a **Simple report** (an office hand-calculation sheet: site, the q_p chain, C_{pe} / W_e per direction, mark-up) with the full working as an optional appendix tick, and a **Detailed report**. Wind produces actions, not a design, so it has no embodied carbon of its own.

Cut & Fill

Topographic survey and ground model, building footprint in true grid coordinates, cut and fill volumes, each excavation solved independently against the ground model, excavation definitions, the prismatic TIN calculation for the selected excavation, end-area sections, and validation. Figures: sections, plan with survey, contours, excavations and section lines, and the 3D ground model. No embodied carbon.

IFC viewer

A take-off report: BIM quantities and A1 to A5 carbon, with material and element-category quantities, the carbon working (quantity × factor), A1-A3 factors from IStructE Table 2.5 and the data points assigned by element type, A4 transport scenarios (Table 2.7), A5 construction process and A5.3 waste (Equation 2.4), and a model comparison where several models are loaded. Charts: life-cycle modules, the bounded A1-A5 range, mass and CO_2e by category and material, the SCORS rating. The status block says TAKE-OFF.

Sketch-ET (preliminary scheme)

The concept sizing report: preliminary layout, 3D model and elevation as on screen, portal frames, beam types, mass and embodied carbon by element, and the SCORS rating. The scheme is concept sizing, not a design, and the combined report labels it that way. Sketch-ET reports can be set to paper size (the A3 landscape default) and orientation; the page size is written into the document and figure limits scale to the sheet.

14.6 The combined (node map) report

From the node map, the Report button opens "Select designs for report". It lists every design on the map, grouped in this fixed order (the same order the chapters follow): Prelim scheme, Wind, Slab, Beam, Column, Base plate, Foundation, Cut & Fill, IFC model. The load path is read top-down: slabs load beams, beams load columns, columns load base plates and foundations, and the IFC take-off closes the report. Sketch-ET leads because it is the concept the detailed designs come from.

- All items are ticked by default. Tick a group to tick all of it, or untick single designs.
- For each ticked foundation the picker first draws the foundation's own Detailed-report figures (asynchronously, with a "Preparing figures" message), then closes. Figures and section bodies are kept, stamped with the inputs and results they were made from, and a stale set is rebuilt in the background and the open report redraws when it arrives.
- The report has one **complete chapter per design**: the design's views, its calculation sections, and its embodied carbon table and charts stay together. The foundation chapters carry the foundation's input data, summary, working calculations, pile reaction schedule (pile caps) and carbon charts from its Detailed report.
- Chapter 1 is Project Details, chapter 2 is **Summary of Designs** (one row per design with the governing check, utilisation and result, plus metric cards: designs included, checks, governing design, total A1-A5 carbon), then one chapter per design, then **Embodied Carbon Summary** and **References and Design Basis**.

- The closing carbon table has one row per design, priced at the A1-A5 figure printed in that design's own chapter, so it cannot disagree with the chapters above it. Where a chapter prints no A1-A5 line (wind, cut and fill), the node-map carbon inventory stands in. If a chapter's figure differs from the node-map panel, the row says so. A donut chart of A1-A5 by design is drawn only when no design has negative carbon (a timber sequestration factor can make a total negative) and the legend merges the smallest designs into "Other designs" beyond four.
- The overall status is PASS only when every design with checks passes; the governing line names the worst utilisation across the selection. A selection of take-off items only is shown as TAKE-OFF ONLY.
- References are the union of every included design's references, de-duplicated.
- If a selected design cannot be generated the report is still produced for the rest and the metric card says how many could not be generated. If none can, the report says "No report could be generated for the selected designs".

14.7 Tips and caveats

- Press Regenerate (or just reopen the window) after changing a design in another tab; the preview redraws itself while it is open but the file you downloaded earlier is a snapshot.
- The preview is text. The figures that go into the Word and PDF file appear under it ("Report figures (as exported)" on the foundation, "Views and design diagrams" on other modules). Check them there before exporting.
- If a figure is missing, the corresponding view could not be drawn. Open the module's view first (some captures, such as Sketch-ET's plan and 3D, use the on-screen view and print a hint if it is not open) and regenerate.
- PDF export relies on the browser's print dialogue. Choose "Save as PDF" as the destination and turn off the browser's own headers and footers if they appear.
- The .doc file is HTML with Office markup. Word will open it, but it is not a native .docx; if your office needs .docx, open it in Word and save as .docx.
- A reinforced-concrete wording check: if you see "plain concrete" in a foundation note you did not expect, reinforcement design is off for that foundation (a new pad starts with it off).
- Sample project names, addresses and file names should not be pasted into the free-text scope box if the report is to be shared as an example; they print as typed.

References. The standards, guidance and data sources used in this chapter are listed, with web links, in Chapter 17.

CHAPTER 15

Checking Your Results

Civil ET is a design aid. This chapter explains what that means for the way you use its results.

15.1 Engineer's responsibility and independent checking

Civil ET produces calculations quickly, but it does not replace the judgement of the engineer who is responsible for the design. Whatever the screen or the report shows, the responsibility for the result stays with you.

- The responsible engineer must check every input, assumption, load path, code clause and result used on a project, and must carry out independent checks (hand calculations, a second program or a peer review) in line with their own quality procedures.
- Civil ET reports carry a disclaimer to that effect. A report is not a certified calculation.
- Some modules are marked beta or alpha in the application (for example Sketch-ET and Cut & Fill). Treat their output as preliminary.
- Where Civil ET and another program disagree, the code clause decides, not either program.
- The calculations use the UK National Annex values. Other National Annexes are not covered.
- Civil ET does not judge whether a design is sensible. It checks the numbers you give it against the code.

CHAPTER 16

Release Notes

These notes summarise what changed in Civil ET, newest version first, grouped by area. The product version is shown in the app's About card and is printed on every report.

16.1 Version 1.2.0

Version 1.2.0 is a large release. It adds new ways of working on the node map, a worked example to start from, and a long list of calculation and report improvements across every module.

Highlights

- **Sticky notes.** A new node for typed text, pen or finger ink and photos, in several colours. Notes can be folded and pinned to cards or groups. A note carries no data and cannot be wired.
- **Example workspace.** Offered on first run, and available later from File > Open the example workspace, so you can see a connected set of nodes working before you build your own.
- **RIBA stages.** Switching stage saves the stage you leave. A new stage starts as a copy of the current one, and saved stages can be reopened.
- **Live load sets.** A load set updates every downstream node as you type, tick boxes or choose presets.
- **Disclaimer in every report.**
- **Touch and pen support** for tablets, with smooth panning, pinch zoom and glide on the node map.
- **Faster node map.** Smoother zoom and pan, simplified tiles when zoomed far out, off-screen cards not drawn, faster start-up.

Node map and workspace

- Thin wires with a live-sync plug, groups, snapping, wire bends, curved or straight wire style and zoom to fit.
- Global undo and redo with Ctrl+Z and Ctrl+Y.
- A floating "?" help panel on every module, and a guided demo tour.
- One-line module headers.
- A node is not re-designed when the numbers it receives have not changed, so an unchanged wire never triggers a foundation redesign.
- Searches for alternative sections run only when you ask for them.
- A utilisation of 0 counts as a pass.
- A safeguard at start-up: if a saved workspace cannot load, an error box offers a backup download instead of leaving a blank screen.
- Feedback can be e-mailed to the team from the app.

Foundation

- Combined footing, pile cap and strip footing calculation fixes.
- STR expression shown on all foundation types, one pile-spacing rule, and no silent input defaults.
- Pile cap section drawing and bar call-outs, and clearer black and white detail drawings with no overlapping labels.
- Foundation views upgraded.

Column and base plate

- Steel and concrete column audits, including signed axial load, frame buckling, class 4 sections and the slenderness basis.
- Base plate: collapsible input panel in the same style as the Column, shear transfer by friction plus anchors, and a number of corrections.

- Psi factors for the imposed-load category used by the column and base plate, including categories E13 and E15 to E19.

Beam

- Concrete beam: crack control (EC2 7.3.3), minimum nominal cover (EC2 4.4.1, UK NA, BS 8500-1), end anchorage and curtailment checks.
- Member self-weight switch. Saved beams keep their existing loads.
- Benchmark hand calculations used to check the beam results.

Slab

- Per-edge bearing shear, the UK NA 5.1.3 option (c) bay check and checkerboard pattern, secondary steel, creep and shrinkage.
- Point-load punching shear (EC2 6.4) and per-support one-way shear.
- EN 1992-1-2 Table 5.8 for the two-way column case.
- Cracked long-term deflection feeds the verdict when it governs.

Durability and cover (all concrete nodes)

- The BS 8500 exposure class drives the minimum concrete cover.
- Design working life input, with EC2 Tables 4.3N and 4.4N.
- User-defined cover is allowed, and durability shortfalls show "Review required".
- Cover information buttons on every concrete node.

Wind designer

- Distance to shore taken from Ordnance Survey high-water data.
- Funnelling per wall, including on IFC and drawn outlines.
- Google Maps site view and wind flow, address search, and a north arrow in the 3D view.
- Reworked plan view and report with a worked calculation.

Sketch-ET

- UK site location with a Google 3D map, grid reference card (10 and 6 figure, E/N, latitude and longitude, DMS).
- Per-change undo, a foldable ribbon for small screens and a floating address search.
- Editing shortcuts and selection fixes.

IFC viewer and Cut & Fill

- Cut & Fill carries an "alpha" label in this version.
- Embodied-carbon and report improvements in the IFC viewer.

Reports

- Every report can be saved as PDF as well as Word.
- Figures are zoomed out until every label fits the picture, labels no longer collide, and a number is kept on one line with its unit.
- Report figures and working calculations were added or corrected across foundation, column, base plate, beam, slab, IFC, Cut & Fill, wind, Sketch-ET and the combined node-map report.
- Typography improvements: units, superscripts and Greek letters.

Appearance

- Contrast fixes in the themes, a "Reading comfort" theme, white text on blue and brown fills, the column 3D view follows the theme, and un-bolded node map text.

- Updated About card text and contacts, and clearer demo tour wording.

16.2 Earlier versions

Versions 1.0.0 and 1.1.0 introduced the Foundation designer, the node workspace and the Column, Base plate, Beam, Slab, Steel sections, Loads, Dimensions, Math, IFC viewer, Cut & Fill, Wind designer and Sketch-ET nodes, embodied-carbon reporting and combined reports. Version 1.2.0 builds on all of these.

16.3 Notes

- Version numbers are shown in the app and on every report. Please quote the version when you report a problem.
- Features marked Beta or Alpha in the app are still being developed and should be used with extra care.

CHAPTER 17

References and Data Sources

This chapter lists the design standards, guidance documents and data sources that Civil ET uses, with the chapters where each one applies and, where one exists, a website for further information. Civil ET does not reproduce the text of any standard. You need your own copy of each standard to check a clause.

Websites are given for reference only. Standards are published under copyright and are normally bought from the publisher or national standards body.

17.1 Design standards (Eurocodes and British Standards)

All Eurocode checks use the **UK National Annexes** unless a chapter says otherwise. Other National Annexes are not implemented.

Standard	Title	Used in
BS EN 1990	Eurocode: Basis of structural design (load combinations, Exp. 6.10)	Chapters 3, 4, 6, 8 and 10
BS EN 1991-1-1	Eurocode 1: Actions on structures, Part 1-1: densities, self-weight and imposed loads	Chapters 4, 6, 7, 8 and 10
BS EN 1991-1-4:2005+A1:2010	Eurocode 1: Actions on structures, Part 1-4: wind actions	Chapter 9
BS EN 1992-1-1:2004+A1:2014	Eurocode 2: Design of concrete structures, Part 1-1: general rules and rules for buildings	Chapters 3, 4, 5, 6 and 8
BS EN 1992-1-2	Eurocode 2: Structural fire design (tabulated method, Table 5.8)	Chapters 4, 6 and 8
BS EN 1992-4	Eurocode 2: Design of fastenings for use in concrete	Chapter 5
BS EN 1993-1-1:2005+A1:2014	Eurocode 3: Design of steel structures, Part 1-1: general rules and rules for buildings	Chapters 4, 5, 6 and 10
BS EN 1993-1-5	Eurocode 3: Plated structural elements (web bearing)	Chapters 4 and 6
BS EN 1993-1-8:2005	Eurocode 3: Design of joints (component method, Table 6.7)	Chapter 5
BS EN 1997-1	Eurocode 7: Geotechnical design, Part 1: general rules (Design Approach 1)	Chapter 3
UK National Annexes	The National Annex to each Eurocode above (for EN 1992-1-1: NA+A2:2014)	All design chapters
BS 8500-1	Concrete: complementary British Standard to BS EN 206, Part 1 (exposure classes and minimum cover, Table A.4)	Chapters 3, 4, 6 and 8

Where to find them:

- **BSI Knowledge** (UK standards, the Eurocodes and their National Annexes): https://knowledge.bsigroup.com
- **European Commission Joint Research Centre** (Eurocodes background, designers' guides and worked examples): https://eurocodes.jrc.ec.europa.eu

17.2 Anchors and fastenings

Document	Used in
ETAG 001 Annex C, Guideline for European Technical Approval of metal anchors for use in concrete (design methods for anchorages)	Chapter 5
EOTA TR 029, Design of bonded anchors	Chapter 5

The European Organisation for Technical Assessment (EOTA) publishes these documents: https://www.eota.eu

17.3 Steel design guidance

Document	Used in
SCI / Steel for Life **Blue Book** (section properties and member resistances), together with SCI P363	Chapters 4, 6 and 7
SCI P364, *Steel Building Design: Worked Examples, Open Sections* (steel beam design workflow)	Chapter 6
SCI P398 (the base plate hand-check route)	Chapter 5
SCI P399 (restraint and C1 values for portal frame checks)	Chapter 10
SCI P360 (tension-flange restraint credit and destabilising loading)	Chapter 6
NCCI documents SN008 and SN045 (frame-mode column effective lengths)	Chapter 4

- **Steel for Life Blue Book** (online): https://www.steelforlifebluebook.co.uk. The section data in Civil ET were imported from Blue Book exports and are not looked up online. Check against the current Blue Book before relying on them.
- **The Steel Construction Institute (SCI)**, publications and NCCI documents: https://www.steel-sci.com

17.4 Concrete, geotechnical and loading guidance

Document	Used in
NHBC Standards, Chapter 4.2 *Building near trees* (tree influence and minimum foundation depth)	Chapter 3
Portal-frame design charts (Weller-style closed-form portal statics)	Chapter 10

- **NHBC Standards**: https://www.nhbc.co.uk
- **The Concrete Centre** (Eurocode 2 guidance and worked examples used as references for the column and slab checks): https://www.concretecentre.com

17.5 Embodied carbon

Document	Used in
IStructE, *How to calculate embodied carbon* (3rd edition as used by the app), Table 2.5 recommended factors, A1 to A5 life-cycle modules	Chapters 4, 6, 8, 10 and 11
SCORS, the structural carbon rating scheme, rated against gross internal area (GIA)	Chapters 10 and 11
RICS whole life carbon assessment benchmarks, used for the A5 demolition and site-activity default rates (35 and 40 $kgCO_2e/m^2$ for all disciplines, with 50 % attributed to structures)	Chapter 11

- **Institution of Structural Engineers**: https://www.istructe.org
- **RICS**: https://www.rics.org

Carbon figures are only as good as the factor you select. Check the edition and the factor against the current guidance, and use project-specific environmental product declaration (EPD) data where you have it.

17.6 Ordnance Survey and other data sources

Data	Used for	Chapter
OS Boundary-Line, mean high water line	Distance from the site to the shore in the wind designer, built into the app	9
OS National Grid (EPSG:27700)	10 and 6 figure grid references, easting/northing, and the coordinate system label on Cut & Fill	10 and 12

Data	Used for	Chapter
Google Maps Platform (aerial, street, 3D map, Places, Elevation)	Site map views, address search and ground levels, when enabled	9 and 10
Esri World Imagery	Aerial map when the Google key is not available	10
Open-Meteo	Elevation fallback	9 and 10
OpenStreetMap / Photon	Address search fallback	10
buildingSMART IFC	The model format read by the IFC viewer	11

- **Ordnance Survey**: Boundary-Line and other open data are available from the OS Data Hub, https://osdatahub.os.uk. Boundary-Line product page: https://www.ordnancesurvey.co.uk/products/boundary-line
- OS data is used under the **Open Government Licence**: https://www.nationalarchives.gov.uk/doc/open-government-licence/version/3/. The attribution *Contains OS data © Crown copyright and database right* is shown in the app where OS data is used.
- **EPSG:27700** (British National Grid): https://epsg.io/27700
- **Google Maps Platform**: https://developers.google.com/maps (use of the maps is subject to Google's terms).
- **Esri**: https://www.esri.com
- **Open-Meteo**: https://open-meteo.com
- **OpenStreetMap** data are © OpenStreetMap contributors: https://www.openstreetmap.org/copyright
- **buildingSMART International**, the IFC standard: https://www.buildingsmart.org

17.7 Notes on use

- The standards listed are those Civil ET's calculations use. A module may implement only part of a standard, and the relevant chapter says which parts.
- Editions and amendments change. Always check the current edition, the National Annex and any published corrigenda before relying on a result.
- Civil ET is a design aid, and the responsible engineer must check the results independently (see Chapter 15).

APPENDIX A

Quick answers: how do I...?

Step-by-step answers to the questions users ask most. Each one names the section that explains it in full. Use the Index at the back, or search the PDF, to find a task by its key words.

A.1 Workspace, node map, projects and reports

How do I add a node to the node map?

1. Find the node you want in the **node library** on the left of the node map. Use the search box to filter by name, tag or node type if needed.
2. Click the entry to drop the card near the middle of what is on screen, or drag the entry onto the map to place it where you drop it.
3. To put the card straight into a group, drop it on the group's frame.

Note: A click nudges each new card a little so several clicks do not stack exactly. See Section 2.2.

How do I add a second beam, slab or foundation design?

1. Remember that every card of a kind (foundation, column, base plate, beam, slab) is a view of one workspace design list, so dropping another card does not create another design.
2. Use the **+ Add** button on the card to create a new design.

Note: The first card of a kind added to a map that has none starts that design list again at one design, and the status line says so. See Section 2.2.

How do I load an IFC model onto the map?

1. Drag an `.ifc` file from your computer and drop it anywhere on the node map.
2. A new IFC viewer node is created at the drop point and the file is loaded into it.

Note: Any other file type dropped on the map is refused with the warning "Only .ifc files can be dropped on the node map". Only the first IFC card is drawn; further models are added to that card's model list. See Section 2.2.

How do I connect two nodes with a wire?

1. Check the pair is allowed (for example Loads to Slab, Slab to Beam, Beam to Column, Column to Foundation).
2. Drag from the **OUT** socket or OUT dot of the source card.
3. Release on the **IN** socket or IN dot of the target card.
4. Read the status line to confirm the link; the wire label names the pair, for example "BEAM REACTIONS -> COLUMN".

Note: Anything not on the allowed list is refused with "Invalid link for this node pair". A node cannot connect to itself and sticky notes take no wires. See Sections 1.6 and 2.3.

How do I remove a wire?

1. Pull the wire off its input, or
2. Release a drag from an OUT socket in empty space to delink the latest wire, or
3. Click the dot on the wire to open its menu and choose **Delete connection**.

Note: To remove every wire at once use **Clear links** (this also removes the locked, transferred loads from the beam designs). See Section 2.3.

How do I remove all wires at once?

1. Find the **Auto sync / Clear links** strip on the map.

2. Click **Clear links**.

Note: This also removes the locked, transferred loads from the beam designs. See Section 2.3.

How do I bend or reshape a wire?

1. Drag the dot in the middle of the wire.
2. On a curved wire this bends the curve; on a straight wire it moves the right-angle bend.

See Section 2.3.

How do I switch between curved and straight wires?

1. Find the wire style switch at the bottom of the map.
2. Choose **Curved** (bezier) or **Straight** (right-angle bends).

Note: The choice is remembered. See Section 2.3.

How do I send a column's load to a particular foundation design?

1. Wire the column to the foundation card.
2. Click the dot on the wire to open the "Connection transfer options" box.
3. Choose which foundation design receives the data (all designs, or one named design such as PF2) and which fields are transferred.
4. Check the preview table, then press **Apply / check transfer**.

Note: Several column outputs can feed different foundation designs at the same time. The base plate has its own "Base plate load transfer" box. See Section 2.3.

How do I feed two values into a Math node?

1. Wire the first source to operand A.
2. Wire the second source to operand B.

Note: Each operand takes one wire only. A second wire to an occupied operand is refused with "delink it first (one wire per input)". See Section 2.3.

How do I turn Auto sync on or off?

1. Find the **Auto sync: ON / OFF** button on the map.
2. Click it to switch the master live-sync setting.
3. When it is OFF, new wires connect but carry no data until you press Sync.

Note: The default is ON and the setting is remembered. See Section 2.3.

How do I pause or resume one wire?

1. Find the **plug** beside the wire's dot.
2. Click the plug to pause or resume that wire's live sync.

Note: A paused wire keeps the data of its last sync and is drawn dashed; a live wire is a solid colour. See Sections 2.3 and 2.15.

How do I push data through a wire manually?

1. Click the ↻ button in the header of the card.
2. The node and everything wired to it syncs now, even through a paused wire or with Auto sync off.

See Section 2.3.

How do I edit a value that arrives by wire?

1. Notice that wired-in loads are locked in the input panel, and that a "Node link" chip in the module header says which node supplies them.
2. Unplug the wire or delete it.
3. Edit the value by hand.

See Sections 1.5 and 2.3.

How do I rename a node?

1. Click the ✎ button in the card header, or double-click the title.
2. Type the new name.
3. Press Enter to commit or Escape to cancel.

Note: Beam and slab cards keep fixed titles ("Beam analysis and design", "Slab analysis and design"). See Section 2.2.

How do I minimise or expand a card?

1. Click ▾ in the card header to minimise the card to its header (the output dots stay).
2. Click ▸ to expand it again.

Note: Collapsing unused cards helps a slow map. See Sections 2.2 and 2.15.

How do I move a card and line it up with others?

1. Drag the card by its header.
2. Switch **Snap** on in the zoom strip to line the card up with the edges and centres of nearby cards (dashed guides show) or to land it on a 20 px grid.
3. To line up several cards, select them, right-click and choose **Align tops of selected**.

Note: Snap affects dragging only. See Section 2.4.

How do I select several nodes?

1. Drag a box across empty map with the left button, or shift-drag.
2. A selection bar appears with **Delete selected** and **Clear**.
3. Press Escape to clear the selection.

See Section 2.4.

How do I group nodes?

1. Select the nodes you want to keep together.
2. Right-click and choose **Group selected nodes…**.
3. To change a group later, right-click it and choose Rename, Add or Edit comment, Select the group's nodes, or Ungroup.
4. To take a card out of a group, right-click it and choose **Remove from group**.

Note: A group has a name, a comment and a colour, moves as one, and is covered by undo. A node in a group can still be wired to nodes outside it. See Section 2.4.

How do I delete a node?

1. Click the × on the card, or select the node and press Delete or Backspace (not while typing in a box).
2. To delete several nodes, select them and use **Delete selected**; Civil ET asks once.

Note: All wires on the node go too. Deleting a Foundation, Column or Base plate card hides it rather than destroying it; drag a fresh one back from the node library. Undo brings a deleted node back with its design. See Sections 2.4 and 2.5.

How do I bring back a deleted Foundation, Column or Base plate card?

1. Open the node library.
2. Drag the entry (or click it) to put the card back on the map.

Note: If a wire needs a hidden core card as an endpoint, the card returns automatically and the status line tells you. See Section 2.2.

How do I zoom, pan and fit the map?

1. Zoom with the mouse wheel (10 % per notch, around the pointer), a trackpad scroll or pinch, or the **-** and **+** buttons (25 % per step).
2. Click **100** to reset to 100 %.
3. Click **Fit**, or double-click empty map, to zoom to fit every node.
4. Pan by dragging with the middle mouse button.

Note: Left-drag on empty map draws a selection box, not a pan. Zoom runs from 1 % to 220 % and the view is saved. Below 30 % zoom cards are drawn as simple tiles. See Sections 2.6 and 2.15.

How do I use the node map with a finger or pen?

1. Use one finger on a card or control as a normal click (double-tap for double-click).
2. Pan the map with one finger on empty map; pinch with two fingers to zoom.
3. Hold for about half a second to right-click (context menus).
4. Hold, then drag on the map to sweep a selection; hold and lift on empty map to open the map menu.
5. Double-tap empty map to zoom to fit.

Note: There is no tablet mode switch; touch works alongside the mouse. See Section 2.6.

How do I undo and redo?

1. Press **Ctrl+Z** to undo (Cmd on a Mac).
2. Press **Ctrl+Y** or **Ctrl+Shift+Z** to redo.
3. Read the small toast, for example "Undo - Beam B1 - 3 more", to see what changed.

Note: Each design has its own history 80 steps deep, so after switching module undo acts on that module's design. History is lost on reload. Cut & Fill and Sketch-ET use their own drawing-level undo. See Section 2.8.

How do I add and use a sticky note?

1. Add the **Sticky note** node from the node library.
2. Type straight onto the paper ("Type a note...").
3. To write by hand, use an Apple Pencil or stylus, or turn the pencil toggle on to write with a finger or mouse. Choose one of three ink colours; use undo-last-stroke or clear-all as needed.
4. Add a picture with the paperclip button, paste, or drop an image on the paper (up to 6 per note).
5. Choose a colour: yellow, pink, green, blue or orange.
6. Drag the corner grip to resize.

Note: The note carries no design data and takes no wires. A picture is refused with a message if browser storage is full. See Section 2.11.

How do I pin a sticky note to a card or group?

1. Drag the note and let go over a card; it sticks (a pin replaces the tape) and travels with that card.
2. Let go over a group frame to make it join the group.
3. To free a pinned note, click the pin button.

See Section 2.11.

How do I fold a sticky note?

1. Click ▾ on the note to fold it into a small dog-eared square showing icons for text, ink and picture count.
2. Click the square to open it again.

See Section 2.11.

How do I save my project and open it again?

1. Open the **File** menu in the header.
2. Choose **Save**. In Chrome or Edge this writes back to the file you opened or last saved; in other browsers it stores a copy in browser storage and tells you to use **Save as...**.
3. To keep a file on disk in any browser, choose **Save as... (.json)**, which downloads the project.
4. To reopen, choose **Open project...** and pick the file.

Note: Open reloads the page. Save a file before using Open, as it replaces what is in the browser rather than merging. Everything is also auto-saved to your browser's local storage, but a project file is how you keep, move or share a job. See Section 2.9.

How do I make a copy of my project?

1. Open the **File** menu.
2. Choose **Duplicate / Copy**.
3. A copy downloads with " (Copy)" added to the project name and today's date.

See Section 2.9.

How do I start a new project?

1. Save your current project first.
2. Open the **File** menu and choose **New project**.
3. Confirm. The foundation designs, project details and related settings reset to defaults.

Note: This resets the foundation side only; delete or clear the node map separately if you want an empty map. See Section 2.9.

How do I get the example workspace back?

1. Save your project to a file first if you want to keep it.
2. Open the **File** menu and choose **Open the example workspace...**.
3. Confirm when asked. The page reloads on the worked example.

Note: This replaces the node map and every design in this browser; your project details and saved stages are kept. A browser that has never run Civil ET opens on the example automatically. See Sections 1.4 and 2.10.

How do I fill in the project details?

1. On the node map, use the project bar across the top: RIBA stage, job number, project name, designed by, reviewed by, client and date.
2. For the full form, including notes, open **File > Project info & metadata**.

Note: These details stamp every report and the saved file name, which is built from RIBA stage, job number, project name and version. See Sections 1.4, 2.9 and 2.13.

How do I change the RIBA stage?

1. Pick a stage from the RIBA stage drop-down in the header or project bar (stages 0 to 7).
2. Civil ET saves the stage you are leaving in full.

3. If the target stage already has a snapshot it opens as it was (the page reloads). If not, the new stage starts as a copy of the stage you left, and edits belong to it alone.

Note: The project-info form and File menu behave identically. The default for a new project is Stage 2. See Section 2.13.

How do I create a custom RIBA stage?

1. Choose **+ Custom stage...** in the RIBA stage drop-down.
2. Enter text such as "Stage 4.5 - Tender / Contractor Design".

Note: A bare number is expanded to "Stage n - Custom Stage". Custom stages are saved with the project. See Section 2.13.

How do I save, advance to or load a RIBA stage snapshot?

1. Open the **File** menu.
2. Choose **Save stage snapshot** to store the current stage in the session, then use **Save as...** to persist it to disk.
3. Choose **Advance to next RIBA stage** to move to the next stage.
4. Pick a saved stage from the list to load it, or use its delete cross to remove it.

Note: Loading the stage you are already in asks before discarding changes made since it was saved. At the end of the list you see "Already at final RIBA stage". See Section 2.13.

How do I change the theme, font and text size?

1. Use the **Theme** control in the header: Dark Navy, Light Slate, B/W, Warm Sand, or Reading comfort (with a Cream, Pale blue or Pale peach tint).
2. Use the **Font** control to choose from the long list; "Aptos / Report style" is first.
3. Use **A-** and **A+** to scale the large display text from -6 to +2.

See Sections 1.5 and 2.13.

How do I take the guided tour?

1. Click the amber **Demo** button in the header for the whole tour (card, node library, title bar, carbon panel, wiring, groups and notes).
2. For a single card, click the small triangle in its header; it tours that card and then the workspace around it.

See Sections 1.4 and 2.14.

How do I send feedback?

1. Click the speech-bubble button: one on the node map for the whole app, one on each card (Section 2.2), and one in each module.
2. Write your feedback and send.
3. If it cannot be sent, the dialog offers **Copy** and **Download .json**, and your mail application may open with the team addresses.

See Sections 1.5 and 2.14.

How do I see the version and contact details?

1. Click the **i** button in the header to open the About card.
2. It shows the version, a one-line description, the verify-your-results notice and contact addresses.

See Section 1.5.

How do I generate a report for one design?

1. Open the module (foundation, column, base plate, beam, slab, IFC viewer, Cut & Fill, wind designer or Sketch-ET).

2. Fill in your project details first so the cover is complete.
3. Click **Report** in the header.
4. Choose a **Report colour scheme** if wanted: Amber engineering (default), Black / white, Corporate blue or Reading comfort.
5. Check the figures listed under the preview ("Views and design diagrams") before exporting.

Note: In the foundation designer you can also choose a Report mode (Detailed or Simple report) and add a **Calc description / scope**, and the button is disabled while a calculation is pending or has failed. If the report cannot be generated, an alert asks you to check the module inputs. See the Reports chapter, "Producing a report".

How do I export a report as Word, PDF or text?

1. Open the report window with **Report**.
2. Click **Word (.doc)** to download a Word document, **PDF** to open the print dialogue and choose Save as PDF, **.txt** to download plain text, or **Copy** to copy the text note to the clipboard.
3. Each button redraws the report first, so the export is the design as it is now.

Note: The Word file is not a native .docx; open it in Word and save as .docx if you need that. In the print dialogue, choose "Save as PDF" and turn off the browser's own headers and footers if they appear. See the Reports chapter, "Producing a report" and "Tips and caveats".

How do I refresh a report after changing my design?

1. Click **Regenerate** in the report window.
2. The preview also redraws itself shortly after you edit the design while the window is open.

Note: A file you downloaded earlier is a snapshot and does not update. See the Reports chapter, "Live data".

How do I produce a combined report of several designs?

1. On the node map, click **Report**.
2. In "Select designs for report", all designs are ticked by default. Untick single designs, or tick or untick a whole group.
3. Wait for the "Preparing figures" message if a foundation is included.
4. Review and export as for a single report.

Note: The order is Prelim scheme, Wind, Slab, Beam, Column, Base plate, Foundation, Cut & Fill, IFC model. Each design gets a complete chapter, followed by an Embodied Carbon Summary and References and Design Basis. The status is PASS only when every design with checks passes. If a design cannot be generated the rest are still produced. See the Reports chapter, "The combined (node map) report".

How do I use the carbon panel and SCORS rating?

1. Open the node map and look at the workspace summary in the right panel.
2. Read the material quantities and the SCORS rating against the gross internal area (GIA).
3. Type a GIA value if you know it; otherwise it is derived from included Sketch-ET designs, or 1000 m^2 is used and the panel says so.
4. Use the tick boxes in the element carbon split to include or exclude elements from both carbon and quantities.
5. Read the quantity and embodied CO_2e charts and the RIBA-stage trend.

Note: With nothing on the map, every total stays at zero until a design node is added. See Sections 1.5 and 2.7.

How do I switch a foundation header between one foundation's carbon and the total?

1. In the foundation module header, click the **Carbon** button.
2. It toggles between the current foundation's embodied carbon and the all-foundation total.

See Section 1.5.

How do I recover when Civil ET fails to start?

1. Read the "Civil ET failed to start" panel, which shows the error text.
2. Click **Download backup** to save everything held in the browser for this site.
3. Click **Clear saved state and reload** to download the same backup and then clear local storage and reload.
4. Or click **Dismiss** to close the panel.
5. Send the backup and the error text to the support addresses on the About card if you need help recovering a job.

Note: The backup is not a project file, so keep your project files as your real archive. See Sections 1.4 and 2.16.

How do I return to the node map from a module?

1. Click the small return-arrow button ("Minimise to blocks") in the module header, or the **Node map** button in the workspace bar.
2. Everything is kept.

See Section 1.5.

How do I open a module from its card?

1. Double-click the card (beam, slab, IFC, Cut & Fill, wind, Sketch-ET), or use the card's open button (foundation, column, base plate).
2. Use the design picker in the module header to move between designs.

Note: Steel sections, Loads, Dimensions, Math and Sticky note are worked entirely on the card. See Section 1.5.

How do I work with load sets?

1. Add a Loads node.
2. Add a data set tab with **+**; rename by double-click or the pencil; delete with **×** (click twice to confirm).
3. Add rows for point, line or area loads by class (dead, live, horizontal) and choose an imposed-load category preset if wanted.
4. Wire the Loads node to a slab, beam, foundation or Math node.

Note: With Auto sync on, a live wire and the node map open, the downstream cards update as you type; presets, class changes, ticks and adding or deleting rows sync at once. See Section 2.12.

A.2 Foundations, columns and base plates

How do I open the Foundation designer?

1. On the node map, add a Foundation node from the node library (the card is titled **Foundation**, tag EC7 / EC2).
2. Double-click the card to open the designer.
3. Alternatively, open one of the designs listed on the card (PF1, PF2 and so on); each is an input point and opens the designer on that foundation.

See Chapter 3, Opening the designer.

How do I choose the foundation type (pad, combined, strip, pile cap)?

1. In the header, use the foundation-system selector.
2. Pick **Pad foundation (PF)**, **Combined footing (CF)**, **Strip footing (SF)** or **Pile cap (PC)**.
3. Enter the inputs on the left; the views and results update for the chosen type.

Note: **Raft foundation (RF)** is listed but reads "in development, locked" and cannot be chosen. See Chapter 3, Overview.

How do I design a pad foundation for a column load?

1. Choose **Pad foundation (PF)** in the header selector.
2. Set the pad length (X), breadth (Y), thickness and depth of soil over the pad, and the column size and offset.
3. On the **Bearing Capacity** card, enter the allowable bearing pressure (SBC) from your site investigation.
4. Enter the loads on the **Superimposed Dead Load (SDL)**, **Live Load (LL)** and **Wind Load (WL)** cards.
5. Read the summary table on the right for PASS or FAIL on each check.

Note: All loads start at zero, so a new pad is designed only for the loads you enter. See Chapter 3, Inputs - pad.

How do I enter the soil bearing capacity (SBC)?

1. Open the **Bearing Capacity** card.
2. Type the allowable bearing pressure in kN/m^2 (the default is 250).
3. Press the **guide** button beside the SBC if you want a table of typical values, from dense gravel above 600 kN/m^2 down to soft clays below 75 kN/m^2.

Note: The tool compares your SBC with the pressure on the base. It does not calculate a bearing resistance from c, phi and unit weight. See Chapter 3, Bearing Capacity card.

How do I use a net bearing allowance?

1. Open the **Bearing Capacity** card.
2. Switch **Net bearing allowance** on, so overburden relief is added to the SBC and the SBC is read as a net value.
3. Leave it off unless your ground investigation report gives a net allowable pressure.

See Chapter 3, Bearing Capacity card.

How do I enter the water table level?

1. Open the **Materials** card.
2. Type the depth in **Water below GL** (m). A value of 0 means no water table is recorded.
3. Choose the **Unit weight of water** (9.81, 10.00 or 10.30 kN/m^3).

Note: Once a level is entered, the pressures that drive the checks use buoyant weights below the water table. See Chapter 3, Materials card and Groundwater.

How do I set the soil parameters for sliding and settlement?

1. Open the **Materials** card.
2. Enter the soil friction angle (default 30 degrees), which feeds the sliding check.
3. Enter the unit weight of soil (default 18 kN/m^3) and the soil modulus Es in MPa (default 20) for the settlement estimate.
4. Choose the interface friction: 1.00 for cast against soil, or 0.67 for precast or smooth.

See Chapter 3, Materials card.

How do I enter column loads on a foundation?

1. Find the load cards: **Superimposed Dead Load (SDL)**, **Live Load (LL)** and **Wind Load (WL)**.
2. In each, enter horizontal X and Y and vertical Z in kN, and the moments Mx and My at the column base in kNm.
3. Use positive Z for downward compression and negative Z for uplift.
4. Add permanent and variable surcharge in kN/m^2 on the **Surcharge Loads** card if needed.

Note: Do not include the base's own weight or the soil over it; the tool adds them to every combination. See Chapter 3, Load cards.

How do I switch a load group in or out of the checks?

1. Look at the bar under the plan or section card.
2. Press the **Use load** buttons for SDL, LL or WL.
3. A switched-out group keeps its values but is ignored in every check.

See Chapter 3, Load cards.

How do I reorder or collapse the input cards?

1. Collapse any left-hand card using its header.
2. Drag the load cards to put them in a new order.

See Chapter 3, Screen layout.

How do I choose between correlated and independent load actions?

1. Open the **Bearing Capacity** card.
2. In **Action components**, pick **Correlated G / Q / W** (the default) or **Independent component envelope**.
3. Use the independent option if you want a more conservative supplementary envelope.

Note: The independent envelope is stated as a scope note in the report, not raised as a failure. See Chapter 3, Action basis.

How do I choose the STR combination expression and punching options?

1. Open the **Punching Shear & Calculation Options** card.
2. Set **STR combination expression** to 6.10, or to 6.10a/6.10b.
3. Choose whether the punching moment is reduced by the soil reaction (the default) or is the full column moment.
4. Set a minimum punching factor beta of 1.00, 1.15 (internal), 1.40 (edge) or 1.50 (corner) if you want a higher floor.

Note: The combination expression affects the structural design only; the geotechnical checks stay on 6.10. See Chapter 3, Punching Shear and Calculation Options card.

How do I read the summary of checks and utilisation?

1. Look at the summary table at the top of the calculation column.
2. Each row gives the design value, the resistance, the utilisation and PASS or FAIL.
3. Press the **?** or **where** link on a row to jump to the card holding its working.
4. Check the overall verdict in the header, summary and report cover.

See Chapter 3, Outputs.

How do I see the full working for a check?

1. Go to the right-hand calculation column.
2. Open the card for the check (Bearing Pressure, Punching, Bending Reinforcement, Top Reinforcement, Sliding/Overturning/Uplift, Settlement).
3. Expand the **Working** section on that card.

See Chapter 3, Screen layout.

How do I turn on reinforced design for a pad?

1. Open the **Materials** card.
2. Switch **Reinforcement design** on (new pads start with it off).

3. Fill in the **Reinforcement** card that appears.

Note: With it off, the pad is designed as plain concrete to EC2 12.9.3 and shows a thickness check instead of punching, shear and crack checks. See Chapter 3, Plain concrete or reinforced.

How do I set the reinforcement bars or mesh?

1. With **Reinforcement design** on, open the **Reinforcement** card.
2. Choose the reinforcement type: bar or mesh.
3. For bars, set diameter and spacing for bottom X and Y and top X and Y (defaults 25 / 150 bottom and 12 / 150 top).
4. For mesh, choose a bottom and top BS 4483 mesh (defaults A393 and A252).
5. Choose the reinforcement grade on the **Materials** card (B500A, B500B, B500C, B450C or B400).

Note: A bar size, spacing or fyk that had to be substituted is named on the panel and in the report, and fails the design. See Chapter 3, Reinforcement card.

How do I set the crack-width limit and aggregate size?

1. Open the **Reinforcement** card.
2. Enter the crack-width limit wmax in mm (default 0.3).
3. Choose the aggregate size (10, 14, 20, 32 or 40 mm); it sets the minimum clear bar spacing in the detailing check.

See Chapter 3, Reinforcement card.

How do I add top reinforcement?

1. Open the **Materials** card.
2. Switch **Top reinforcement** on, for hogging, uplift or lift-off.
3. Set the top bars on the **Reinforcement** card.
4. Read the **Top Reinforcement** card for the result.

See Chapter 3, Materials card and Reinforced design.

How do I set the exposure class and cover?

1. Open the **Materials** card.
2. Choose the **Exposure class (BS 8500-1)** (default XC2, wet, rarely dry).
3. Choose what the bottom and the sides are cast against (formwork, blinding, or directly on soil).
4. Leave **Cover mode** on auto to use the minimum nominal cover from BS 8500 plus EC2 4.4.1, or switch to user-defined to type your own covers.

Note: A user cover that falls short of the minimum is flagged "review", not "fail". See Chapter 3, Materials card.

How do I set the concrete grade and design working life?

1. Open the **Materials** card.
2. Pick the **Concrete grade** (default C30/37) and the unit weight of concrete (24, 25 or 26 kN/m^3).
3. Choose the **Design working life** (50 or 100 years).
4. Set **Special QC of concrete production** to yes if it applies; this gives one structural class lower.

See Chapter 3, Materials card.

How do I design a combined footing?

1. Choose **Combined footing (CF)** in the header selector.
2. Enter the base size, then each column's size, position and G, Q and W loads.
3. Leave the column position lock on if you want the columns to move when you resize the base.
4. Edit dimension chains between columns, or lock individual dimensions, as needed.

5. Read the results in the summary and the check cards.

Note: You can start from one of the five presets; converting a pad starts from "Chapel double column". Column data for combined footings is entered manually. See Chapter 3, Combined footing (CF).

How do I design a strip footing?

1. Choose **Strip footing (SF)** in the header selector.
2. Enter the strip width, the wall width, the eccentricity of the wall on the strip and the wall loads per metre (G, Q, W vertical, horizontal and moment).
3. Leave **Reinforcement design** off for a plain concrete strip, or switch it on for flexure, minimum steel, one-way shear and crack width.

Note: The defaults are 1.0 m by 0.6 m, 1.0 m thick, C28/35 and SBC 123 kN/m^2. See Chapter 3, Strip footing (SF).

How do I design a pile cap?

1. Choose **Pile cap (PC)** in the header selector.
2. Use the pile count selector to pick 2, 3, 4, 5, 6, 7 or 8 piles.
3. Enter pile diameter, spacing, edge clearance, embedment, pile length, positional tolerance and the allowable compression, tension and lateral loads.
4. For a 3-pile cap, pick triangular (strut-and-tie) or rectangular (beam theory).
5. Enter the loads and read the pile checks and cap design results.

Note: Switching template replaces the cap geometry, pile coordinates and reinforcement, but keeps the loads and any linked column. See Chapter 3, Pile cap (PC).

How do I check pile reactions without designing the cap reinforcement?

1. Choose **Pile cap (PC)**.
2. Switch **Reinforcement design** off on the **Materials** card ("Analysis only").
3. The tool reports pile reactions and the EC7 pile checks without designing the cap.

See Chapter 3, Pile cap (PC).

How do I find alternative designs with lower carbon?

1. Scroll to the **Alternative designs** card.
2. Press **Find reinforced alternatives** (or **Find mass concrete alternatives** when reinforcement design is off).
3. Wait for the search; a combined footing search can take 5 to 15 seconds.
4. Click a card to apply its size, depth and bars in one update.
5. Press **Search again from the current design** to repeat.

Note: The search never runs on an input change, and results clear when any input other than plan size, depth, bars or column positions changes. Cards are ranked lowest carbon first and each passes every check. See Chapter 3, Alternative designs.

How do I check the NHBC tree-influence founding depth?

1. Open the bottom drawer and select the **NHBC / Trees** tab.
2. Enter the plasticity index (class or modified), tree species, distance and height.
3. Read the required founding depth from NHBC Chapter 4.2 (Table 4b and Table 5).
4. The pad passes when its thickness plus depth of soil is at least the required depth.

Note: The tab is disabled for pile caps. See Chapter 3, Other inputs and Tips and known caveats.

How do I attach a soil report?

1. Open the bottom drawer and select the **Soil Report** tab.
2. Attach the ground investigation PDF.
3. Type the values it gives (SBC, phi, E, unit weight) into the "from report" boxes.
4. Apply them to write them into the inputs.

Note: The tool does not read the values from the PDF for you. See Chapter 3, Other inputs.

How do I set embodied carbon options?

1. Open the carbon panel in the right drawer.
2. Choose the cement type, and the concrete and steel EPD source.
3. Set the A4 transport and A5 construction/waste percentages (defaults 12 % and 4 % of A1-A3).
4. Read the A1 to A5 totals in the panel.

See Chapter 3, Other inputs.

How do I add project details to reports?

1. Open the project file menu in the header.
2. Choose **Project info & metadata**.
3. Enter project name, number, designer, checker, client, date and RIBA stage once.
4. They then appear on every report.

See Chapter 3, Other inputs.

How do I switch between plan, section and 3D views?

1. In the middle column, each of the two view cards has a drop-down.
2. Choose **Plan View**, **Section View** or **3D Viewer**.
3. On the section view, choose the X-X or Y-Y cut.
4. Use the toolbar toggles on the plan to show or hide loads, reinforcement, punching zone, trees and the eccentricity resultant.

See Chapter 3, Views and the detail drawing.

How do I see the black and white detail drawing?

1. On the plan card, press the **B/W detail** button.
2. Choose the X-X or Y-Y section.
3. Use the **q** toggle for corner pressures and the **Reinf** toggle for reinforcement.
4. Read the bar marks (B1/B2 bottom, T1/T2 top) and the bar bending schedule band on the drawing.

Note: The drawing uses the bar layout the calculation uses, so the drawn bars match the calculation. For pile caps it includes a pile table. See Chapter 3, Views and the detail drawing.

How do I get the bar bending schedule?

1. Open the **B/W detail** drawing for a reinforced design.
2. Read the schedule band, which lists mark, shape, diameter, length and number with BS 8666 rounding.
3. The schedule also appears in the report.

See Chapter 3, Outputs and Views and the detail drawing.

How do I produce a foundation report?

1. Press the **Report** button in the header.

2. Choose **Report mode**: **Detailed report** or **Simple report**.
3. Choose a colour scheme: Amber engineering, Black / white, Corporate blue or Reading comfort.
4. Type an optional calc description or scope for the report header.
5. Use **Regenerate**, **Copy**, **.txt**, **Word (.doc)** or **PDF** (choose Save as PDF in the print dialogue).

Note: The button is disabled while a calculation is pending or has failed. See Chapter 3, Reports.

How do I add another foundation design?

1. Use the Foundation node card to add a design; it copies the active design's inputs.
2. Each design (PF1, PF2 and so on) appears as its own input point with its type and worst utilisation.
3. Open the one you want from the card or from the header.

See Chapter 3, Opening the designer and Multiple foundations and the Foundation node.

How do I link a column to a foundation?

1. Draw a wire from a Column node's OUT socket to a Foundation input point (one pad or pile cap, or the main "all eligible" input).
2. The column's loads, size, material, section, grade and base plate size are written to the foundation.
3. The load boxes driven by the link are locked and show a note saying so.

Note: Only pad foundations and pile caps accept incoming data; combined and strip footings stay manual. See Chapter 3, Column to Foundation wiring.

How do I stop a foundation taking data from the column?

1. In the foundation header, turn off the **Use column node** toggle.
2. Alternatively, delink the wire or switch off that transfer field on the wire.
3. Your earlier manual loads are remembered so you can return to them.

See Chapter 3, Column to Foundation wiring.

How do I read the worst utilisation on the Foundation card?

1. Look at the foundation row on the node card.
2. The percentage is the worst utilisation across every gating check.
3. Green is up to 85 %, amber is above 85 %, red is above 100 %.

See Chapter 3, Opening the designer and Multiple foundations and the Foundation node.

How do I undo a change in the foundation designer?

1. Press Ctrl+Z to undo and Ctrl+Y to redo; history is kept per design.
2. Text boxes keep their own native undo.

See Chapter 3, Tips and known caveats.

How do I open the Column node and switch between steel and concrete?

1. Open the Column node from the map (add it from the node library if needed, tag EC3 / EC2).
2. In **Column details**, set **Design material** to steel or concrete.
3. The rest of the panel changes to match.

See Section 4.1.

How do I add several columns to one Column node?

1. Open the Column node.

2. Use the design rows (C1, C2, ...) on the node card; each has its own input and output dots.
3. Use the header picker to switch the active design.

See Section 4.1.

How do I enter column loads by case?

1. In the **Design loading** card, leave **Enter loads by case (DL / LL / WL)** switched on.
2. Fill the grid of dead, imposed and wind rows for P/Z, Vx, Vy, Mx and My.
3. Use positive values for compression and negative for tension.

See Section 4.2.1.

How do I enter a single set of factored design loads on a column?

1. In **Design loading**, switch **Enter loads by case (DL / LL / WL)** off.
2. Type NEd, VEd (y-y), VEd (z-z) and the bottom moments about y-y and z-z.
3. Enter factored ULS values; no partial factors are added.

See Section 4.2.1.

How do I enter the column top moments?

1. The grid's Mx and My rows are the bottom moments.
2. Type the top moments in **Bending moment top - about y-y / z-z (kNm)**.
3. Use unredistributed elastic values.

Note: For steel, give both ends; leaving the top at 0 treats the column as having a single-ended moment. See Section 4.2.1 and Section 4.9.

How do I let Civil ET apply the load factors to my column loads?

1. With the grid on, set **The row values are** to **characteristic (unfactored)**.
2. Choose the imposed-load category (default B).
3. The tool checks every consistent load set from Eurocode expression 6.10 and reports the governing set.

Note: Choose **already factored** (the default) if your numbers already include the factors. See Section 4.2.2.

How do I feed a column from a beam, slab, IFC or Sketch-ET node?

1. Draw a wire from the beam, slab, IFC viewer or Sketch-ET node to the Column node.
2. The column switches to characteristic rows automatically and locks the selector.
3. Rows written by the wire show a lock.

Note: Several beams on one column are summed. A slab sends its area loads times a quarter-panel tributary area, so adjust the grid if your tributary differs. See Section 4.2.3.

How do I type my own loads on a wired column?

1. Select **Type my own loads instead** on the column.
2. The wire stays connected and still delivers geometry and section.
3. Select **Use the model's loads again** to hand control back.

See Section 4.2.3.

How do I choose a steel column section and grade?

1. Set **Design material** to steel.
2. Choose the family (default Universal columns (UC)) and the section (default 203x203x46).

3. Choose the steel grade (default S355), or pick user defined and enter fy and fu.
4. Set hollow forming (hot finished or cold formed) and I fabrication (rolled or welded) if relevant.

Note: A connected Steel sections node overrides family and section. See Section 4.3.1.

How do I set steel column effective lengths and buckling?

1. Enter the system length in mm (default 3500).
2. For each axis, choose a buckling mode: factor (Lcr = K times L), frame, or direct (type Lcr).
3. Use K factor presets: both ends fixed 0.5, one end fixed 0.7, restrained 0.85, pinned 1.0, cantilever 2.0.
4. Switch **Sway** on for the relevant axis if the column buckles in a sway mode.
5. Set KT for the torsional buckling length factor if needed.

See Section 4.3.2.

How do I use frame mode for steel buckling lengths?

1. Set the buckling mode to **frame** for the axis.
2. Enter the lengths, and optionally the sections, of the columns and beams above and below.
3. A beam counts only once a section is picked, and a missing length means no member.
4. A foundation base is taken as nominally rigid.

See Section 4.3.2.

How do I set the lateral-torsional buckling moment shape?

1. Find the LTB moment shape setting.
2. Choose **auto** (from the end moments), **uniform** (conservative, C1 = 1.0) or **custom** (type your own C1, 1.0 to 2.8).

See Section 4.3.2.

How do I read the steel column checks?

1. Read the summary card rows: shear y-y, shear z-z, axial compression, axial tension, class 4 scope, bending y-y, bending z-z, biaxial bending, buckling in compression, buckling in bending and combined buckling.
2. Each row shows action, resistance, a utilisation bar and PASS or FAIL.
3. Open the **Working calculations** card for the intermediate values.

Note: The column passes when the governing utilisation is at most 1.00 and the section is not an unsupported class 4. See Section 4.3.3.

How do I find a lighter or lower-carbon steel section?

1. Open the **Alternative steel sections - carbon optimised** card.
2. Press its run button.
3. Choose an option and apply it.

Note: The search runs only when you press the button; a note says "inputs changed" when results are out of date. See Section 4.3.1 and Section 4.7.

How do I design a reinforced concrete column?

1. Set **Design material** to concrete.
2. Edit the section size and the bar arrangement on the section view (default 300 x 300 mm, C30/37, 4 bars of 20 mm, links 8 mm at 175 mm).
3. Set the reinforcement grade, exposure class (default XC3/XC4), design working life and aggregate size.
4. Enter the loads and effective lengths.

5. Read the summary rows and the **Working calculations** card.

Note: RC columns are rectangular only, with a perimeter bar arrangement. See Section 4.4.1.

How do I set the RC column effective length and braced or unbraced behaviour?

1. Switch **Braced** on (default) or off for unbraced (sway).
2. For each axis choose a mode: factor (default 0.70), stiffness (relative flexibilities k1 and k2, default 0.10) or direct (type l0).
3. Set the end conditions for Y and Z (default fixed-fixed).

See Section 4.4.1.

How do I choose the slenderness and imperfection basis for an RC column?

1. Open the slenderness and imperfection basis setting (default conservative).
2. Choose conservative or refined.
3. Read the **Slenderness and second order effects** card for the result.

Note: The conservative basis uses simplified imperfections about both axes; the refined basis uses the single most unfavourable direction. See Section 4.4.2.

How do I set cover for an RC column?

1. Leave **Cover from exposure / fire** on for automatic nominal cover (minimum plus 10 mm, rounded up to 5 mm).
2. Switch it off to type the cover yourself.
3. Optionally enter a cmin,dur override (0 means automatic).

Note: A user cover below the bond or fire minimum fails; below only the durability value gives "Review required". See Section 4.4.1 and Section 4.4.2.

How do I set fire resistance for an RC column?

1. Choose the fire resistance: Not checked, R30, R60 (default), R90, R120, R180 or R240.
2. Set the fire load ratio (default 0.7) and the fire effective length (intermediate floor, top floor, or normal-temperature).
3. Read the fire resistance summary row.

Note: If the validity conditions are exceeded the row shows "CHECK B" and fails until the design is changed or checked by Method B or Annex C outside the tool. See Section 4.4.1 and Section 4.4.2.

How do I find an optimised RC column size and bars?

1. Open the **Optimised column options** card.
2. Press its run button.
3. Choose and apply an option.

See Section 4.4.2 and Section 4.7.

How do I view the RC column interaction diagram and 3D view?

1. Open the **N-M interaction diagram** card to see the section's envelope with the design points.
2. Open the **Column section + 3D views** card for live reinforcement.
3. Open **Concrete column properties** for section values.

See Section 4.4.2.

How do I get a column report?

1. Open the Report from the column workspace.

2. Choose Word, PDF (opens the print dialogue; choose Save as PDF), copy or text.
3. Choose one of four colour schemes (Amber engineering, Black / white, Corporate blue, Reading comfort).
4. Press Regenerate to re-read the live design.

See Section 4.6.

How do I set the column quantity and carbon factors?

1. Set the quantity of identical columns (default 1).
2. Choose carbon factors, or enter custom factors or EPD file fields.
3. A4 defaults to 12 % and A5 to 4 % of A1-A3.
4. For RC, switch **Take-off rebar from the design** off to use a flat percentage allowance for carbon only.

See Section 4.4.1 and Section 4.5.

How do I send column loads to a foundation?

1. Draw a wire from the Column node to a pad or pile cap Foundation input.
2. On the wire, switch off any item you do not want sent (P, Vx, Vy, Mx, My, size, section, utilisation, carbon).
3. Choose one foundation or all eligible ones as the target.
4. Use the wire's manual sync button if you need to force a re-send.

Note: The column's own weight is added to the dead axial load it sends; the foundation applies its own factors. See Section 4.8.

How do I send column loads to a base plate?

1. Draw a wire from the Column node to the Base plate node.
2. Use the wire menu to choose the cases (DL, LL, WL, SW), the shear source (Vx, Vy or resultant) and the moment source (Mx, My or resultant).
3. Add manual add-ons for N, V or M if required.

See Section 4.8 and Section 5.7.

How do I open a base plate design and add more?

1. Open the Base plate node from the map (add it from the library if needed, tag EC3 / 1-8).
2. Each design (BP1, BP2, ...) has its own dots and utilisation on the node card.
3. Use the header picker to switch designs.

See Section 5.1.

How do I choose the base plate calculation style?

1. In the **Design forces** card, find **Calculation basis**.
2. Choose **EN 1993-1-8 detailed** (default) or **SCI P398 simplified**.
3. Use the simplified basis for a transparent hand-check.

See Section 5.2.

How do I enter base plate design forces by hand?

1. In **Design forces**, leave **Use column node data** off.
2. Enter NEd (kN), VEd (kN) and MEd (kNm).
3. Use the **Include** switches for NEd, VEd and MEd to isolate individual checks.

Note: The loads are used as final design actions; no partial factors are applied. A negative NEd is treated as zero. See Section 5.3.1.

How do I take base plate forces from a Column node?

1. Switch **Use column node data** on and draw the Column to Base plate wire.
2. The force fields are disabled and show a summary line ("DL+LL+WL -> N, V, M").
3. Press **Restore manual** to return to the numbers you typed before linking.

See Section 5.3.1 and Section 5.7.

How do I set the column and weld on a base plate?

1. In **Column details**, choose the column section (default 305x305x118) and steel grade (default S355).
2. Enter the column eccentricity on the plate if any.
3. Choose the weld leg size (6, 8, 10, 12 or 14 mm).

Note: When linked to a Column node, the section is disabled and taken from the column. See Section 5.3.2.

How do I set the base plate size and grade?

1. In **Base plate details**, enter plate length, width and thickness (defaults 600, 600 and 40 mm; thickness 10 to 150 mm).
2. Choose the plate grade (default S275), or user defined and enter fy and fu.

See Section 5.3.3.

How do I include or leave out friction in the shear check?

1. In **Base plate details**, set the friction coefficient (default 0.20).
2. Switch **Include friction check** off, or set the coefficient to 0, when the anchors are to carry all the shear.

See Section 5.3.3 and Section 5.8.

How do I lay out the anchors?

1. In **Anchor layout**, choose the anchor diameter (12 to 36 mm) and the bolt hole clearance.
2. Set the number of bolts in each row on the left and right sides (default 2 and 2).
3. Enter the edge distances x and y for each side.

Note: ETAG design is limited to 3 bolts per side; more is flagged and forces a fail. See Section 5.3.4.

How do I run a full anchor check to ETAG 001 / TR029?

1. In **Anchor layout**, switch **Design anchors using ETAG 001 / TR029** on.
2. Fill in the **ETAG 001 / TR029 anchor data** card: anchor type (Bonded or Mechanical), fyk, fuk, effective anchorage depth, splitting and bond data.
3. Choose Cracked or Non-cracked concrete, the installation factors and the combined interaction exponent.
4. Read the anchor tension, anchor shear and combined anchor rows.

Note: Cracked is the default; choose Non-cracked only where justified. Check the product data against its European Technical Assessment. See Section 5.3.6 and Section 5.8.

How do I design anchors as ordinary bolts?

1. In **Anchor layout**, switch **Design anchors using ETAG 001 / TR029** off.
2. The anchors are then treated as ordinary bolts using their diameter.
3. Read the bolt group tension and bolt shear rows.

See Section 5.3.6 and Section 5.5.

How do I set the foundation under a base plate?

1. In **Foundation / concrete**, enter foundation length, width and thickness (defaults 3000, 3000 and 700 mm).
2. Choose the concrete class (C20/25 to C40/50; default C25/30).
3. Or switch **Use foundation node data** on and wire a Foundation to the Base plate to take these values.

See Section 5.3.5 and Section 5.7.

How do I read the base plate results?

1. Read the summary card: Moment (or Axial), Shear transfer, weld rows and, where relevant, anchor rows.
2. Check the governing check name and utilisation in the header and on the node card.
3. Open the working cards for every intermediate figure.

Note: Tension rows appear only when the anchors are actually in tension. See Section 5.5.

How do I get a base plate report?

1. Open the Report from the node toolbar.
2. Choose Word, PDF via the print dialogue, copy or text.
3. Choose one of four colour schemes.

See Section 5.6.

How do I send base plate size and utilisation to the foundation?

1. Draw a wire from the Base plate node to the Foundation.
2. Plate size, column section text, utilisation and carbon move across.

See Section 5.7.

A.3 Beams, slabs, loads and steel sections

How do I add a Beam node and open the full workspace?

1. Find **Beam analysis** (tag "M · V · δ") in the node library.
2. Click it, or drag it onto the map.
3. **Double-click** the card, but not on a button, box or drop-down.
4. The workspace opens with the input panel on the left, the member view and diagrams in the centre, and quantities and embodied carbon on the right.

Note: A new beam is a 6 m simply supported concrete beam, 300 × 500 mm, C30/37. See Section 6.1.

How do I keep several beam designs in one Beam node?

1. Open the Beam workspace.
2. Use the picker in the module header to add a design (B1, B2, B3 and so on).
3. Use the same picker to rename, switch or delete designs.
4. Each design has its own output dot on the node card and its own report.

Note: A wire leaving the card names the design it comes from. See Section 6.1.

How do I analyse a continuous beam with several spans?

1. In the **Tools** card pick **Support**.
2. Click the member at each position where a support is needed. Any number of supports is allowed.
3. Select each support and set its type to **pinned**, **roller** or **fixed**, or set its position with the slider or typed box in the **Span and load sliders** card.

4. Add loads with the other tools, then read the **V**, **M** and deflection diagrams.

Note: The beam needs two supports at different positions, or one fixed support, otherwise it is reported as a mechanism. See Section 6.2.1.

How do I change the span of a beam?

1. Open the **Span and load sliders** card.
2. Move the span slider or type the span in metres (limits 0.1 to 999, default 6).
3. Check the support positions, which stay where you put them.

See Section 6.2.1.

How do I add a point load, UDL or moment to a beam?

1. In the **Tools** card choose **Point load**, **UDL / diagonal** or **Moment**.
2. Click the member where the load should go.
3. Set the magnitude and position with the slider or typed box in the **Span and load sliders** card. A UDL has a start and end magnitude, so it can be trapezoidal.
4. Set the class of each load to **G** (permanent) or **Q** (variable).

See Section 6.2.1.

How do I move or delete a support or load on a beam?

1. Choose **Select / move** and drag the support or load along the member.
2. To remove one, choose **Delete** and click it.
3. Press **Esc** at any time to hand the pointer back to **Select / move**.

Note: Loads that arrived by wire are locked bands and cannot be dragged or edited by hand. See Sections 6.1 and 6.2.1.

How do I snap support positions to a grid or draw an angled beam?

1. Set the grid snap to 0, 0.25 or 0.5 m (default 0.25) for support positions.
2. Set the angle snap to 0, 15, 30, 45 or 90 degrees to draw the member at an angle.
3. The load direction is projected onto the member normal, so gravity always loads the beam downward.

See Section 6.2.1.

How do I fill a beam UDL from a dead-load or imposed-load table?

1. Select a UDL.
2. Pick an entry from the **Gk table** (dead loads) or the **Qk table** (EN 1991-1-1 imposed categories).
3. Enter the loaded width. The box starts at 1.00 m and is never invented silently.
4. The UDL becomes the area load multiplied by the width.

See Section 6.2.1.

How do I undo or redo a change in a beam?

1. Press **Ctrl+Z** to undo and **Ctrl+Y** to redo.
2. This covers the whole beam design, not only the drawn geometry.

See Section 6.1.

How do I design the beam about its minor axis?

1. Use the bending-axis switch on the drawing and choose the **minor** axis.

2. The analysis inertia, section modulus, shear area and width all change accordingly.

Note: A steel beam bent about its minor axis cannot buckle laterally and its web does not classify. See Section 6.2.3.

How do I include shear deformation in a beam analysis?

1. Open the **Analysis** card.
2. Tick **Timoshenko / shear**. It is off by default.
3. Leave it off to match a hand calculation or on to line up with frame packages that include shear deformation.

Note: The effect matters most on short deep members and transfer beams. Non-linear and Vibration are locked. See Section 6.3.

How do I override the stiffness of a beam?

1. Find the member strip beside the drawing.
2. Tick **own value** and enter your own stiffness.

Note: Otherwise EI comes from the material: concrete uses the grade and the b × h section, steel uses the selected Blue Book section, and timber uses 11 GPa. See Section 6.2.2.

How do I choose the EN 1990 load combinations for a beam?

1. Open the **Load combinations** card.
2. Set **Loading mode** to **Characteristic Gk / Qk - EN 1990 combinations** (the default for new beams) so that the node factors each load.
3. Set the permanent factor to 1.35 for Exp. (6.10), or 1.25 for the worse of 6.10a and 6.10b.
4. Choose the **ψ load category** (default B, office).

Note: Class each load G or Q correctly, because they are factored differently. See Sections 6.4.1 to 6.4.3.

How do I use pre-factored loads on a beam?

1. In the **Load combinations** card set **Loading mode** to **Pre-factored design loads (γ = 1.0)**.
2. Enter the loads exactly as you want them applied for both ULS and SLS.

Note: This legacy mode has no separate quasi-permanent set. Self-weight still takes 1.35 at ULS. See Section 6.4.1.

How do I read design actions rather than service actions on a beam?

1. Find the **Analysis read at** switch in the member view header or the **Load combinations** card.
2. Choose **ULS** to see design moments, shears and reactions. The default is **SLS**.
3. For SLS you can pick characteristic, frequent or quasi-permanent.

Note: This changes only what you read. The strength checks always run on the ULS combination. See Section 6.4.4.

How do I include or exclude beam self-weight?

1. Find the **Member self-weight included** switch (on for new beams).
2. Switch it off if you have already entered the self-weight as a load.

Note: Concrete adds b × h × 25 kN/m^3 (3.75 kN/m for 300 × 500). Steel adds the Blue Book mass per metre. Timber adds nothing. See Section 6.4.5.

How do I design a reinforced concrete beam to EC2?

1. In the **Material** card choose **Concrete beam**.
2. Check that the **EC2 concrete design** switch is on (the default).
3. Enter width, depth, concrete grade, fyk, cover, link diameter, bottom bars, top bars and link legs, diameter and spacing.

4. Read the PASS or FAIL chip and the governing check in the header.

Note: With the design switch off you get analysis only, with no checks and no quantities. See Section 6.5.

How do I set bars and links on a concrete beam?

1. Open the concrete beam design cards.
2. Set **Bottom bars** (default 3 × 20) and **Top bars** (default 3 × 25) as number × diameter.
3. Set the shear links: legs, diameter and spacing (default 2, 8 mm, 250 mm).
4. Watch the flexure, shear and spacing checks update.

Note: Bottom and top are the member's sides about the minor axis. See Sections 6.5.1 and 6.5.2.

How do I fix a beam that needs compression steel?

1. Look for the Compression steel (As2) check or the warning about the opposite face.
2. Provide bars on the opposite face at least equal to the required As2, or deepen the section.

Note: The node fails the beam if the compression steel provided is short. See Section 6.5.2.

How do I use moment redistribution or the ductility cap on a beam?

1. Enter the redistribution ratio δ between 0.7 and 1.0.
2. Optionally tick the K' cap of 0.168.

Note: Analysis moments stay elastic. δ only changes K' and the service stress. See Sections 6.5.1 and 6.11.

How do I design the top steel over a pinned end of a monolithic beam?

1. Leave **Monolithic end supports** on (the default).
2. The top face is then designed for the larger of the analysed hogging moment and 0.25 times the adjacent span moment.
3. Switch it off for a beam sitting on bearings, steelwork or masonry.

See Section 6.5.2.

How do I check shear in a concrete beam?

1. Provide links in the shear card.
2. Read the Shear links and Shear strut utilisations.
3. If the strut check fails, increase the section or the concrete class.

Note: The peak shear is used unreduced and no credit is taken for loads near the support. See Section 6.5.3.

How do I set cover, exposure class and cover mode on a concrete beam?

1. Choose the **Exposure class** (default XC1), the design working life (50 or 100 years) and whether special QC applies.
2. Choose **Cover mode**: **auto** takes the computed minimum, **user** keeps your value.
3. Set Δc dev (10, 5 or 0 mm) and what the beam is cast against (formwork, blinding or soil).
4. Optionally enter a cmin,dur override (0 = automatic).

Note: Auto cover passes by construction, so Nominal cover may rank near 100 % without being the real governing check. See Section 6.5.5.

How do I check fire resistance of a concrete beam?

1. Set the **Fire period (R)** to 30, 60, 90, 120, 180 or 240 minutes.
2. Read the Fire check, which compares width and axis distance against the EN 1992-1-2 tables.

3. Set it to 0 to switch the check off.

Note: The simply supported table is used unless there are more than two supports or any fixed support. See Section 6.5.5.

How do I check anchorage, laps and curtailment on a concrete beam?

1. Enter the support width, the support kind (direct or indirect) and the bar end at the support (straight, bend or continuous).
2. To curtail bars, enter the number of curtailed bottom or top bars and their distance from the support.
3. Read the End anchorage and Curtailment checks and the anchorage and lap lengths.

Note: With nothing curtailed, every bar runs the full length. See Section 6.5.5.

How do I check crack control on a concrete beam?

1. Choose the **Crack k basis**: EC2 or conservative.
2. Read the As,min crack check and the Crack control check.
3. For beams 1000 mm deep or more, read the reported skin-reinforcement area.

Note: Side bars are reported, not designed. See Section 6.5.4.

How do I check deflection of a concrete beam?

1. Read the Span/depth check, which uses a K factor taken automatically from the supports. You can force a value.
2. Tick **Brittle partitions** where it applies, since it affects the factor only for spans over 7 m.
3. Read the explicit long-term deflection against span/250. Span/500 is also shown.

See Section 6.5.6.

How do I find a better concrete beam size?

1. Press **Find optimised beam options**.
2. The node searches nearby widths and depths with bar and link choices.
3. Passing designs are listed ranked by A1-A3 embodied carbon.

Note: Nothing is searched until you press the button. If inputs change, the results show an "inputs changed" note and the button returns. See Section 6.5.7.

How do I design a steel beam to EC3?

1. In the **Material** card choose **Steel beam**.
2. Check that the **EC3 steel beam design** switch is on.
3. Choose the **Section family** (Universal beams or Universal columns), the **Section** and the **Steel grade** (S275, S355 or S460).
4. Choose the bending axis and the lateral restraint.
5. Read the utilisation list for Shear, Bending, LTB, Deflection and so on.

Note: The default section is 356x171x45 in S355. See Section 6.6.1.

How do I set lateral restraint on a steel beam?

1. Set **Lateral restraint** to **Fully restrained (no LTB)**, **supports only**, **supports + point-load positions** or **custom positions**.
2. For custom positions, enter the list of x positions in metres.
3. Set **Restrained flange** to both, top only or bottom only.

Note: A point of contraflexure is not a brace and a fixed support is not treated as a minor-axis or torsional restraint. See Sections 6.6.1 and 6.6.2.

How do I set the C1 factor for lateral-torsional buckling?

1. Leave **C1** on auto to fit it to each segment's moment diagram.
2. Or pick a Blue Book row value (1.00, 1.13, 1.35, 1.50, 1.77, 2.00 or 2.50).
3. Or enter a custom value between 1.0 and 2.8.

See Section 6.6.1.

How do I make web bearing count in a steel beam check?

1. Set **Support / load introduction** to **bearing on the bottom flange**.
2. Set the **Stiff bearing length** ss (default 50 mm).

Note: With the default **connection** setting the check is reported but not counted. See Section 6.6.2.

How do I choose the deflection limits for a steel beam?

1. Set the **Deflection limit on variable actions** to auto, 360, 250, 200 or 180.
2. Set the **Total (Gk + Qk) deflection limit** to 200, 250, 360 or off.

Note: Auto uses span/360 for spans and L/180 for cantilevers. Total cantilever limits take half the denominator. See Section 6.6.2.

How do I find a lighter steel section?

1. Press **Find lighter steel sections**.
2. The node designs candidate sections in the chosen family.
3. Passing sections are listed by embodied carbon.

Note: It runs only when you press the button. See Section 6.6.4.

How do I analyse a timber beam?

1. In the **Material** card pick timber (design is shown locked, "design coming").
2. Enter the section b × h (default 100 × 300 mm), density (450 kg/m^3) and E (11 GPa).
3. Read reactions and the V, M and deflection diagrams.

Note: There are no design checks and no automatic self-weight. Only deflection against span/250 is shown. See Section 6.7.

How do I read the quantities and embodied carbon of a beam?

1. Look at the right-hand panel.
2. Set the number of identical beams and the scope: current beam or all beams on the node.
3. Choose the steel, concrete, rebar and timber factors. You can add a custom factor or EPD.
4. Read A1 to A5 carbon, with A4 at 12 % and A5 at 4 % of A1-A3 by default.

Note: If the design is switched off every quantity reads zero and "not assessed". See Section 6.8.

How do I replace the reinforcement quantity with my own bar-bending-schedule number?

1. Open the reinforcement-quantity card.
2. Choose **Auto**, **Rate** (kg per m^3 of concrete, from a preset or your own) or **Total kg** per beam.
3. Compare with the derived value shown on the card.

Note: Beam, slab, column and foundation all use this same control. For a slab the total is in kg per panel. See Section 6.8.

How do I send slab loads to a beam?

1. Draw a wire from the Slab OUT dot ("Slab area loads gk / qk") to the Beam IN dot.
2. On the beam, choose the loaded width: the default is half lx (edge beam), full lx (internal beam) or a typed width, separately for G and Q.
3. For a two-way slab, choose which edge the beam supports; the load is a trapezoid on a long edge and a triangle on a short edge.
4. Optionally tick the option that sets the beam span from the slab edge length.

Note: The slab's point and line loads are not sent, only the area loads. See Sections 6.9.1 and 8.9.2.

How do I send a Loads node, Math result or IFC element to a beam?

1. Draw a wire from the source to the beam.
2. Line rows (kN/m) become full-span UDLs, vertical force rows (kN) become mid-span point loads, and area rows become UDLs through a loaded width starting at 1 m.
3. A Math result is placed by its unit; a plain number becomes a permanent UDL.
4. Wired bands show as locked and tagged with their source.

Note: Any wire that writes loads switches the beam to characteristic mode. See Section 6.9.1.

How do I mute or detach wired loads on a beam?

1. To leave a band out of every solve, switch off ("mute") that band. It stays on screen.
2. To stop wires touching the loads, use **Type my own loads instead** in the **Supports and loads** card. Span, material and section still come across.
3. To remove a band, delete the wire that wrote it.

Note: Deleting a wire removes only the bands that wire wrote. See Section 6.9.1.

How do I send beam reactions to a column?

1. Draw a wire from the beam design's OUT dot ("Support reaction Gk / Qk") to a Column.
2. On the wire choose the support: the governing one by default, or a named support such as S1 or S2.
3. Choose what it becomes on the column: axial P/Z (default), or shear Vx or Vy.
4. Optionally set the connection eccentricity and send the end moment for a fixed support.

Note: Reactions are characteristic and split into G and Q. The column applies its own factors. See Section 6.9.2.

How do I get a calculation note for a beam, slab or other design?

1. Open the report from the module.
2. Choose the format (Word, PDF, copy or text) and a colour scheme.
3. The report re-reads the live design each time.

Note: With a design switched off the report says "analysis only". See Sections 6.10 and 8.7.

How do I add a Steel sections node and pick a section?

1. Add **Steel sections** (tag "Blue Book") from the node library.
2. Choose the **Family**, then the **Section** (for example 356x171x45).
3. Choose the **Data route** to see the matching table.

Note: Changing family resets the section to the first one in the family and the route to its first route. A column in steel mode adds this node for you if none exists. See Sections 7.1.2 and 7.1.3.

How do I see section properties, fire parameters or resistance tables?

1. On the Steel sections card choose the **Data route**.

2. Use **Dimensions and properties** for geometry, areas, moduli and constants.
3. Use **Detailing & fire parameters** for dimensions C, N and n and section factors.
4. Use the grade routes (for example axial compression S355, buckling resistance moment, web bearing) for tabulated resistances.

Note: If a table is not in the data the card says "No ... table found". See Sections 7.1.3 to 7.1.5.

How do I use a Steel sections node to set a column section?

1. Draw a wire from the Steel sections node to the specific Column that should use it.
2. The column takes the family and section from the node and says so.
3. If Auto sync is off, press Sync now to bring changes across.

Note: This node does not feed the Beam node. Choose the beam section on the beam itself. Treat Base plate and Foundation wires as column-only unless you see the section arrive. See Sections 7.1.6 and 7.1.7.

How do I open the Blue Book website from a Steel sections card?

1. Click the **Open Blue Book** link on the card.
2. It opens steelforlifebluebook.co.uk in a new tab.

See Section 7.1.1.

How do I add a Loads node and create a load set?

1. Add **Loads** (tag "Input") from the node library.
2. A new node starts with one set, "Set 1", and no rows.
3. Press **+ Add load** to add a row.
4. Press **+** on the tab strip to add another set.

Note: Only the active tab is sent by wire. See Sections 7.2.1 and 7.2.2.

How do I rename, switch or delete a load set?

1. Click a tab to make it active.
2. Double-click a tab, or use the edit box, to rename it.
3. Click the **x** on a tab, then click again when it asks "sure?", to delete it.

Note: The last set cannot be deleted. See Section 7.2.2.

How do I fill in a row in a Loads node?

1. Type a **Name**, or leave it blank to show the preset name.
2. Choose the **Class**: **G** permanent, **Q** variable or **H** horizontal.
3. Choose the **Direction** (Fz, Fx or Fy) and a **Preset** from the library, or a custom value.
4. Enter the **Value** and any **Dims** (L, B, H in m or mm).
5. Keep the sigma tick on to include the row in the total.

Note: Changing class resets the preset. See Sections 7.2.3 and 7.2.4.

How do I turn an area load into a load per metre or a force in a Loads row?

1. For an area-load row tick **area -> kN** and fill L and B (or enter a direct area) to get kN.
2. Or tick **width -> kN/m** and enter the loaded width to get kN/m.
3. The two ticks are mutually exclusive.

Note: A missing dimension gives a red message such as "enter the loaded width". See Section 7.2.4.

How do I enter a self-weight in a Loads node?

1. Add a **G** row.
2. Choose "Concrete - reinforced" (25 kN/m³) from the bulk densities, or a custom density.
3. Fill L, B and H to get kN, L and B to get kN/m, or one dimension to get kN/m².

Note: There is no separate self-weight input. See Sections 7.2.4 and 7.2.7.

How do I read a Loads answer in newtons instead of kilonewtons?

1. Set the row's **Answer unit** to the N family (N, N/mm, N/mm²).
2. The inputs stay as typed.

See Sections 7.2.3 and 7.2.9.

How do I wire a Loads node to a beam, slab, foundation or Math node?

1. Draw a wire from the Loads output dot to the target.
2. Pick the **DL**, **LL** or **Horizontal** dot for class totals, the **sigma** dot for the total, or a row's own dot for that single answer.
3. For a slab, choose on the wire which slab field the class drives (screed, finishes and services, movable partitions or imposed qk).
4. For a foundation the DL, LL and horizontal vectors fill the DL, LL and WL rows as locked loads.

Note: A Loads node cannot be wired to a Column or Base plate. Row and sigma dots send exactly that answer. See Sections 7.2.5 and 7.2.6.

How do I attach a reference document to a Loads node?

1. Add a PDF or image reference to the card.
2. Images up to 400 kB are stored in the project.
3. PDFs and larger images are stored by name only ("reference only - file not stored").

Note: Keep those files separately. See Section 7.2.8.

How do I use a Dimensions node?

1. Add the Dimensions node.
2. Enter **Name**, **L**, **B** and **Depth D** (all in m, default 1.0).
3. The card shows area (L × B) and volume (L × B × D).
4. Wire the **L**, **B**, **Area** or **Volume** dot to a Math node.

Note: Dimensions can be wired only to a Math node and takes no input. See Sections 7.3.1 to 7.3.3.

How do I do a calculation with the Math node?

1. Add a Math node.
2. Type A or wire a value into it, choose the operator (+, -, ×, /, max or min), then type B or wire a value.
3. Read the **Result** dot.
4. Wire the Result to a Beam, Slab, Foundation or another Math node.

Note: A wired operand is locked; delink the wire to type a number again. Several wires into one operand are summed. See Sections 7.4.1 to 7.4.4.

How do I turn a pressure into a force with Math?

1. Wire a Loads row in kN/m² to operand A.
2. Wire a Dimensions **Area** (m²) to operand B and choose ×.

3. The result is in kN, which a foundation accepts as an axial load.

Note: A total in kN divided by a Dimensions L gives kN/m for a beam. See Section 7.4.4.

How do I design a one-way or two-way slab?

1. Add **Slab design** (tag "EC2") and double-click it.
2. In the **Slab type** card choose **One-way spanning** or **Two-way restrained**.
3. Set the span L, panel width Ly and depth h (defaults 4.8 m, 6.0 m, 175 mm).
4. Set the support width, support or continuity condition (one-way) or panel case (two-way).
5. Read the PASS, WARN or FAIL chip.

Note: Flat slab, Ribbed slab and the ComFlor composite deck are locked in this release. See Sections 8.1 and 8.2.1.

How do I set the effective span of a slab?

1. In **Span measured as** choose "Effective span leff entered directly" or "Support centres - compute leff".
2. With support centres, the node adds the support allowances to the clear span and shows the leff used.

Note: For a two-way slab the shorter side is always lx, so entering a longer L swaps them. See Section 8.2.1.

How do I choose the panel case of a two-way slab?

1. In **Panel case** pick one of the nine cases (for example Interior panel, or Four edges discontinuous).
2. A continuous edge continues into the next panel; a discontinuous one does not.

Note: A ly/lx ratio above 2 triggers a warning that the slab should be treated as one-way. See Section 8.2.1.

How do I enter slab loads?

1. Open the **Loads** card.
2. Accept the self-weight (live from the depth) and edit screed, finishes and services, imposed qk, movable partitions and snow sk.
3. Use the preset lists, or replace any preset with a custom value.
4. Choose the imposed category and the combination factor (1.35 or 1.25).

Note: Snow is a second variable action, and for a Category H roof imposed load and snow are not combined. See Sections 8.2.2 and 8.2.3.

How do I mute a load component on a slab?

1. In the 3D view use the study chips.
2. Mute self-weight, screed, finishes, imposed, partitions or snow.

Note: Muted components are excluded from every calculation and from what the slab sends downstream. See Section 8.2.2.

How do I add point loads and line loads to a slab?

1. Make sure **Plate FE analysis** is on.
2. Press **+ Point load** or **+ Line load**.
3. For a point load enter P (or pick a concentrated load from the library), x, y, loaded width b (default 0.3 m) and class.
4. For a line load enter w, the centre position, direction X or Y, length, width and class.
5. Drag the load in the 3D view if you want.

Note: With the FE off, point and line loads are switched off but kept. See Section 8.2.4.

How do I set materials, cover and fire rating on a slab?

1. Choose the concrete grade (default C30/37) and fyk (500).
2. Choose exposure class, working life and cover mode (automatic or user-defined), and bottom and top cover.
3. Choose the fire rating: REI 60, 90, 120 or 240.

Note: The default for bottom bars anchored at a simple support is 50 %. See Section 8.2.5.

How do I set the reinforcement of a slab?

1. In the **Reinforcement** card set the main bottom bars (default H12 at 175) and, for two-way, the second direction (H12 at 200).
2. Set the top bars over supports (H12 at 175).
3. Set the secondary distribution bars for a one-way slab (H10 at 350).

See Section 8.2.6.

How do I choose the slab analysis basis and mesh density?

1. In the **Analysis** card keep **Plate FE analysis** on.
2. Choose the analysis basis: full plate FE, code coefficients plus plate FE for point and line loads, or code coefficients plus an effective-width strip.
3. Choose the **FE mesh density**: Coarse, Normal (default), Fine or Very fine.

Note: Normal is converged for design. Go coarse for quick load dragging and fine only for small loaded areas or loads close to a support. See Sections 8.2.8 and 8.10.

How do I handle a concentrated load peak in a slab?

1. Read the smoothing warning in the working.
2. Set the **Peak smoothing width** in metres, or leave 0 for automatic.

Note: The warning appears when smoothing removes more than 15 % of the raw peak. See Sections 8.2.8, 8.3.2 and 8.10.

How do I set a one-way slab as an end span with continuity?

1. Choose "end span, continuous end support" (or the other continuity options) under support or continuity.
2. Set **Spans in the run** (2 to 6, default 3).
3. Set the panel side edges to symmetry or free edges.
4. If neighbouring spans differ, enter the percentage so you are warned.

See Sections 8.2.1, 8.2.7 and 8.2.8.

How do I use the 20 % support redistribution option in a slab?

1. Leave **20 % support redistribution** on (the default).
2. Check that Qk/Gk is not more than 1.25, imposed load is not more than 5 kN/m^2, and a one-way bay is over 30 m^2.
3. If these are not met, the node envelopes alternate load arrangements or adds a failed coefficient-validity check.

See Sections 8.2.8 and 8.3.3.

How do I check punching under a point load on a slab?

1. Add a point load.
2. Leave **Punching check under point loads** on.

3. Read the Punching row for each point load.

Note: If it fails, increase the bearing area, the depth or the steel. The node does not design punching links. See Section 8.4.3.

How do I fix a slab deflection failure?

1. Read the Deflection l/d row and the long-term deflection row.
2. Increase the depth or the provided bars.
3. Check the humidity, age at first loading and cement class under the long-term inputs.

Note: The unmodified default slab does not pass its deflection checks. See Sections 8.4.4 and 8.10.

How do I find an economical slab depth and reinforcement?

1. Press **Find optimised sections**.
2. The node tries depths in 25 mm steps and the lightest passing bar arrangement at each.
3. Results are ranked by A1-A3 carbon per m^2.

Note: Flat and ribbed types are not searched. Nothing is searched until you press the button. See Section 8.5.

How do I view the slab in 3D and see a reinforcement heat map?

1. Open the **3D view** tab in the centre.
2. Switch on the heat map for As,req on the bottom or top face, as an envelope or per direction.
3. Drag the loads to move them.

See Sections 8.6 and 8.2.4.

How do I wire a slab to a beam and on to a column?

1. Wire the slab OUT dot to a Beam (area loads become line loads through a loaded width).
2. Wire the beam OUT dot to a Column.
3. Alternatively wire the slab directly to a Column, which takes a quarter-panel tributary area.

Note: Muted loads stay muted downstream. See Sections 8.9.2 and 6.9.2.

How do I feed loads into a slab from a Loads or Math node?

1. Wire a Loads row or Math result in kN/m^2 to the slab.
2. Choose which slab field the load drives. Ticking several fields writes the value to the first and zeroes the others.
3. Driven fields are locked; use the switch in the slab's **Loads** panel to turn incoming wires off and edit by hand.

Note: Deleting a wire releases its fields, setting them to 0 and the preset to custom. See Sections 8.2.2 and 8.9.1.

How do I set quantities and carbon for a slab?

1. In the right-hand panel set the number of identical panels.
2. Choose the carbon scope: this slab or all slabs on the node.
3. Optionally enter your own rebar rate (kg/m^3) or total kg per panel.

Note: The slab uses a rebar allowance (default 110 kg/m^3), not the bars in the layout. See Section 8.8.

A.4 Wind, Sketch-ET, IFC and Cut & Fill

How do I add a Wind designer node and open it?

1. On the node map, add a **Wind designer** node (tag "EC1-1-4 - UK NA").
2. Double-click the node card to open the full-screen module.

3. If you open the Wind module with no wind node in the project, press **+ Add wind case**.
4. To move between several wind designs, use the node switcher in the module header, or its "Add wind" option.

Note: The node card has no OUT socket of its own; it shows a per-design output dot instead. See Section 9.2.

How do I find the wind pressure on a building at a given postcode?

1. In the **Site and wind climate** card, tick **Look up addresses, postcodes and ground level online**.
2. Type the postcode in **Site search** and pick the suggestion. The site is then locked.
3. Check the marker on the **Wind map** tab and read any warning above the stage.
4. Describe the building in the **Plan outline** card (roof, plan size, height).
5. Open the **Results** tab. Sheet 2, Peak velocity pressure, gives the peak velocity pressure, and sheet 4 gives the pressure on every zone.

Note: Postcodes are only looked up online when the tick is on. A postcode point can differ by roughly 17 m from another geocoder's position. See Sections 9.3.1 and 9.6.4.

How do I set the site from an OS grid reference?

1. In **Site search**, type the grid reference, for example `TQ 3003 8039`.
2. Alternatively, type the **Easting** and **Northing** (in km, OS National Grid) directly in the Site and wind climate card.
3. Check the 10-figure grid reference, latitude/longitude, address and postcode shown in the grid reference card.

Note: The grid reference is held to the metre, so you can compare it with an external grid reference finder. See Sections 9.3.1 and 9.5.3.

How do I set the site from latitude and longitude?

1. In **Site search**, type the position as `lat, lon`.
2. Accept the result; the site is placed and locked.
3. Check the marker on the **Wind map** tab.

See Section 9.3.1.

How do I stop the site moving when I click the wind map?

1. In the **Site and wind climate** card, press **Lock** (a site found by search is already locked).
2. A locked site cannot be moved by a stray map click or drag.
3. To move it again, press **Unlock**.

Note: Rotation still works while the site is locked. See Section 9.3.1.

How do I place the site by clicking on the wind map?

1. Make sure the site is **Unlocked**.
2. Open the **Wind map** tab.
3. Pan and zoom, then click where the site is.
4. Check the Easting and Northing in the Site and wind climate card.
5. Press **Lock** once it is placed.

See Sections 9.3.1 and 9.6.1.

How do I set the altitude of the site?

1. In the **Site and wind climate** card, type the **Altitude A** in m above sea level (default 20).
2. Alternatively, tick **Look up addresses, postcodes and ground level online** and the altitude is filled in for you; you can overwrite it.

3. To use the conservative altitude factor at every height, tick **Simple altitude factor**. Leaving it off uses the NA.2b expression above 10 m.

See Section 9.3.1.

How do I enter the basic wind speed myself?

1. In the **Site and wind climate** card, set **Wind climate** to **Enter vb,map directly**.
2. Type the **vb,map** in m/s (the UK range is about 21.5 to 31 m/s).
3. Read the effect on the Results tab, sheet 1.

Note: In manual mode the site position still places the building on the maps and sets the fetch and directions. See Section 9.3.1.

How do I go back to reading the wind speed from the map?

1. In the **Site and wind climate** card, set **Wind climate** to **From the map (Figure NA.1)**.
2. If the card says the marker is off the traced coastline, move the site.

See Section 9.3.1.

How do I set the building roof type?

1. In the **Plan outline** card, choose **Roof**: **Flat**, **Monopitch**, **Duopitch** or **Hipped**.
2. Fill in the **Roof detail** inputs that appear under the roof type.
3. For a pitched roof, check the pitch; choosing a pitched roof with no pitch sets 15 degrees.

See Sections 9.3.2 and 9.3.2.1.

How do I model a monopitch roof?

1. In the **Plan outline** card, set **Roof** to **Monopitch**.
2. In **Roof detail**, choose which wall carries the high eave and type the pitch.
3. Enter the **Height** to the low eave.
4. Read the results: wind onto the low eave is 0 degrees, onto the high eave 180 degrees and along the eaves 90 degrees.

See Sections 9.3.2.1 and 9.5.6.

How do I model a duopitch or hipped roof?

1. In the **Plan outline** card, set **Roof** to **Duopitch** or **Hipped**.
2. In **Roof detail**, choose whether the ridge runs along x or y.
3. Type the pitch. For a hipped roof also type the main pitch and, if different, the hip-end pitch (blank means the same as the main pitch).
4. Enter the **Height** to the eaves.

Note: For a hipped roof the windward face pitch governs. See Section 9.3.2.1.

How do I add a parapet, curved eaves or mansard eaves to a flat roof?

1. Set **Roof** to **Flat**.
2. In **Roof detail**, choose the eaves detail: **Sharp**, **With parapet**, **Curved eaves** or **Mansard eaves**.
3. Type the parapet height, the curved eaves radius ratio or the mansard angle.

Note: A parapet references the roof to the height plus the parapet height. See Section 9.3.2.1.

How do I change the plan size and height of the building?

1. In the **Plan outline** card, choose **Plan shape**: **Rectangle**, **Square** or **Drawn polygon**.

2. Type **Plan size X, Y** and **Height**, or open the **3D and flow** tab and click a dimension box, then type a new value.
3. Height is to the eaves (default 12 m); plan size defaults to 20 by 12 m.

See Section 9.3.2.

How do I rotate the building to match the plot?

1. Type the **Orientation** (degrees clockwise from north to the building +y axis) in the **Plan outline** card.
2. Alternatively use the rotate control on the plan bar.
3. Keep the site locked while you rotate.

Note: Orientation decides which wall faces each wind and the fetch for each face. See Sections 9.3.2 and 9.11.

How do I draw the building outline on the plan?

1. Open the **Plan - zones** tab and press **Draw plan outline**.
2. Click the first corner, then type a length and press Enter (or click) for each side.
3. Use the **90 degrees** chip (or the key O) to switch the 90 degree snap on and off.
4. Type `6<30` for 6 m at 30 degrees, or `@4,3` for an offset.
5. Close the outline with Enter on an empty box, or by clicking the first corner. Backspace undoes the last point; Esc cancels.

Note: Drawing switches the plan shape to a polygon. Press **Back to a rectangle** to return. See Section 9.4.1.

How do I sketch the building over a PDF or image of the plan?

1. Open the **Plan - zones** tab.
2. Switch on the **Underlay** chip and load a PDF page or image.
3. Set the page, scale, offset, opacity and invert controls.
4. Use **Draw plan outline** to trace the building over it.

See Section 9.4.1.

How do I bring in an IFC model to give the building shape?

1. Open the **Other structures** card.
2. Press **Import IFC model** (a plain IFC file only), or **Take from an IFC node** if the project has an IFC node.
3. Check the footprint, height and orientation. They are locked and only the orientation can change.
4. To edit the footprint, press **Release the IFC lock**.
5. In the 3D view, use the **IFC model** chip to show a ghost of the imported mesh.

Note: Only the largest cluster in plan is kept, so separate outbuildings do not distort the footprint. See Section 9.4.2.

How do I set the terrain to town or country?

1. In the **Terrain and exposure** card, choose **Terrain**: **Country** or **Town**.
2. For Town, type the **Distance inside the town boundary** (0.1 to 20 km, default 2).
3. Optionally type the **Height of surroundings** and the **Spacing of obstructions**.

Note: Country uses a town factor of 1.0, which is conservative. Taking the town edge (0.1 km) is the conservative reading. See Section 9.3.4.

How do I measure the distance to the shore automatically?

1. In the **Terrain and exposure** card, keep **Measure the fetch to the shoreline from the map** ticked (the default).
2. Open the **Wind map** tab and check the dashed shore rays and distance pills for each direction.

3. If a narrow tidal river is treated as the shore, open the fold "Shore settings (only if needed)" and raise **Ignore tidal water narrower than** (default 0.5 km).

See Sections 9.3.4 and 9.5.4.

How do I enter the distance to the shore myself?

1. In the **Terrain and exposure** card, untick **Measure the fetch to the shoreline from the map**.
2. Type **Distance upwind to the shoreline** in km (0.1 to 100, default 20).
3. This one value is used for all directions.

See Section 9.3.4.

How do I cap the distance to the shore for Northern Ireland or missed water?

1. Open the fold "Shore settings (only if needed)" in the **Terrain and exposure** card.
2. Type a value in **Shore no further than** (km).

Note: The built-in shore data covers Great Britain only; Northern Ireland falls back to the coastline traced from Figure NA.1. The manual cap is conservative. See Sections 9.5.4 and 9.11.

How do I allow for orography?

1. In the **Terrain and exposure** card, tick **Orography is significant (Figure NA.2)**.
2. Type the **Orography factor co** (default 1.0).

Note: The node does not derive the orography factor from terrain; you supply it. See Sections 9.3.4 and 9.11.

How do I change the season factor or the return period?

1. In the **Wind directions and factors** card, pick the **Season factor** (year, winter, summer or a single month).
2. Choose the **Annual probability of exceedence**: 0.02, 0.01, 0.04, 0.1 or a typed value.

Note: Use a season factor below 1.0 only for temporary works in a known season. 0.02 gives a probability factor of 1.0. See Section 9.3.3.

How do I treat every wind direction as having a directional factor of 1.0?

1. In the **Wind directions and factors** card, tick **Take cdir = 1.0 for every direction**.
2. The peak velocity pressure then becomes one number.

Note: This is conservative. See Section 9.3.3.

How do I add the diagonal wind directions?

1. In the **Wind directions and factors** card, tick **Add the four diagonal directions**.
2. This adds 45, 135, 225 and 315 degrees relative to the building.

Note: The wind directions are not typed; they are the normals to the building's faces, so a rotated building is analysed square-on to each wall. See Section 9.3.3.

How do I allow for funnelling between buildings?

1. Open the **3D and flow** tab.
2. Click the **Funnelling** tag on each wall that faces a neighbouring building; red means on.
3. Read the **Compare** tab for normal against funnelling values.
4. Confirm the gap to the facing building is between e/4 and e, or untick the wall.

Note: Funnelling only changes a wall in the wind cases where it is a side wall. See Section 9.5.7.

How do I choose the internal pressure basis?

1. In the **Openings and internal pressure** card, keep **Include internal pressure cpi** on.
2. Pick the basis: **Unknown openings: +0.2 / -0.3**, **From the openings listed below**, **Openings spread evenly (enter mu)**, **One dominant face (enter its cpe)** or **Enter cpi directly**.
3. Type the value the basis asks for (mu from 0 to 1; or the cpe and area ratio for a dominant face).

Note: The unknown openings basis is the default and the conservative choice. See Section 9.3.6.

How do I list the openings in a building?

1. Choose **From the openings listed below** as the internal pressure basis.
2. Press **+ Opening**, or click a wall on the **Plan - zones** tab and press **+ Opening here**.
3. For each opening set the wall (or Roof), the area in m^2 (default 2) and the offset from the middle of the wall.
4. Read the small table that shows which openings fall in which zone for each direction.

See Section 9.3.6.

How do I switch internal pressure off?

1. In the **Openings and internal pressure** card, turn off **Include internal pressure cpi**.
2. Turn it back on to restore the basis you had chosen.

Note: Acceptable for the overall horizontal force on a closed building, but not for cladding, fixings or roof uplift. See Section 9.3.6.

How do I set the dynamic factor cs cd?

1. In the **Dynamic response cs cd** card, choose a mode: automatic, **cs from Table NA.3 x cd**, **Annex B detailed procedure**, or **Enter cs cd directly**.
2. For the table mode, type cd (1.00 to 1.25, read from Figure NA.9 yourself).
3. For Annex B, enter the construction type, the fundamental frequency and the damping.

Note: In automatic mode the node uses 1.0 and shows a warning if that is not justified. See Section 9.3.7.

How do I include surface friction or a net pressure coefficient?

1. In the **Funnelling and analysis options** card, choose **Surface friction**: **None**, **Smooth**, **Rough** or **Very rough**.
2. To use one net coefficient for the overall force, tick **Use the net pressure coefficient for overall loading**.
3. To change the air density, edit the value (default 1.226 kg/m^3).

Note: Friction is ignored automatically where the code allows. See Section 9.3.5.

How do I split the windward wall into strips for a tall building?

1. In the **Funnelling and analysis options** card, type a **Strip height on the windward wall** in m.
2. Leave it blank for one strip of height b.

Note: A building taller than twice b may be split into strips. See Sections 9.3.5 and 9.5.5.

How do I design a signboard, free-standing wall, tower or mast instead of a building?

1. In the **Other structures** card, change **Structure type** from Building.
2. Choose Tower, mast or chimney; Signboard or billboard; Free-standing wall or fence; or Parapet on a building.
3. Fill in the inputs for that type (for example width, height and clear height for a signboard; height, length, solidity and returns for a wall).

See Section 9.3.8.

How do I see the wind pressure on every wall and roof zone?

1. Open the **Plan - zones** tab.
2. Choose the direction in **Wind from**.
3. Pick a quantity in the **Metric** drop-down: we, we fun, net, cpe, force, area or length.
4. Read the traffic-light zone boxes and the zone table under the drawing.

See Section 9.6.2.

How do I see the wind flow around the building in 3D?

1. Open the **3D and flow** tab.
2. Orbit, pan and zoom the model; press **Reset view** to return to the default camera.
3. Switch on **Wind flow** to see the streamlines.
4. Use **Boxes**, **Labels** and **Zone dims** to show force boxes, zone labels and zone widths.

Note: The north dial turns with the camera; its amber arrow marks where the wind comes from. See Section 9.6.3.

How do I read the full wind calculation step by step?

1. Open the **Results** tab and choose a direction.
2. Read sheets 1 to 8: basic wind velocity, peak velocity pressure, geometry and zone sizes, external pressure on every zone, internal pressure, worst net pressure for cladding and fixings, structural factor and overall force.
3. Scroll to the "All directions" summary and the "Where the numbers come from" table.
4. In the right-hand drawer, the **Site and wind climate** card repeats the chain; press a small **?** on a line to jump to the input that feeds it.

See Sections 9.6.4.

How do I find the overall horizontal wind force and the worst suction?

1. Open the **Results** tab.
2. Read sheet 8, Overall force, for each direction. The governing horizontal case is the largest absolute force.
3. Read sheet 6, Worst net pressure for cladding and fixings, for the worst local pressure and which zone and direction it comes from.
4. Roof uplift per direction is also listed.

See Sections 9.5.9 and 9.7.

How do I compare normal and funnelling pressures?

1. Open the **Compare** tab.
2. Read normal we against funnelling we fun, zone by zone, on one scale.

See Section 9.6.5.

How do I look at the code charts for my design?

1. Open the **Charts** tab.
2. Choose a chart, such as the exposure factor, town correction, peak velocity pressure, directional factor or pressure coefficients.
3. Hover to read values; the amber points are this design.
4. Press a chart's **?** for its help note.

See Section 9.6.6.

How do I show or hide the floating cards in a viewer?

1. In the map, plan or 3D viewer, use that viewer's **Cards** switch.

2. The choice is remembered per viewer in the browser.

See Section 9.2.

How do I view the site on a Google aerial or 3D map?

1. Open the **Overview** or **Wind map** tab.
2. Switch on the Google aerial (2D) or Google photorealistic 3D view.
3. In the **3D and flow** tab, press **3D map** to lay the zones and streamlines over the photorealistic map.

Note: These views need an online connection. See Sections 9.6.1 and 9.6.3.

How do I produce the wind report?

1. Press the **Report** button in the module header.
2. Tick **Include full working (appendix)** if you want the worked calculation.
3. Choose Word or text output as offered.

Note: The appendix gives the governing direction in full and shows only what changes for the others. Views and diagrams follow the working. See Section 9.8.

How do I send the wind force to the Beam, Slab, Column, Foundation or Base plate node?

1. On the node map, drag a wire from the wind design's output dot to the receiving node. The wire is labelled "WIND ACTIONS".
2. The governing horizontal force is passed as two magnitudes, WLx and WLy, in kN.
3. Apply load combinations and factors in the receiving node.

Note: The wind node has no input socket. The load is a characteristic action, and the receiving node applies its own sign convention. See Section 9.10.

How do I use the wind force as a value in a Math node?

1. Wire the wind design's output dot to the **Math** node.
2. Use it as a value source.

See Section 9.10.

How do I check the wind node against my hand calculation?

1. Read each step of the chain on the **Results** tab or the site card beside your own working.
2. Check the usual differences: shortest fetch within the direction sector, the directional factor taken over a 45-degree sector either side of the face normal, h/d for each direction, and rounded map wind speeds.

See Section 9.9.

How do I start a Sketch-ET design?

1. On the node map, add a **Sketch-ET** node (Concept scheme).
2. Select a design row (P1, P2 and so on), rename it, copy it with the row's copy button, or press **+ Add Sketch design** to add another from defaults.
3. Press **Run** on the card (Run P1 or Re-run P1), or **Open Sketch-ET**, which opens the module and runs the design.

Note: A design that has not been run sends no results down its wires. See Section 10.2.

How do I load a PDF, DXF or image of the architect's plan?

1. On the **Underlay and grid** tab, open the Underlay card.
2. Choose **Load a PDF, DXF or image**.
3. For a multi-page PDF, choose the page.

4. To share one sheet across floors, use **Put this sheet behind every level**.

Note: If the PDF cannot be drawn, export the plan as PNG or DXF instead. Keep sheets small because the image is stored with the design. See Section 10.5.1.

How do I set the scale of the underlay?

1. On the Underlay card, use **Calibrate**.
2. Click two points on the drawing and type the real distance.
3. Check a second known dimension before trusting the scale.

See Section 10.5.1.

How do I replace or remove an underlay?

1. On the Underlay card, choose **Replace** or **Remove**.
2. Confirm the prompt.
3. A removed sheet is kept for the session so it can be put back; a replacement keeps the previous scale and position.

See Section 10.5.1.

How do I set up the column grid?

1. On the **Underlay and grid** tab, either draw lines with the Grid line tool or use the **Setting-out grid** card.
2. In the card, set the bay spacings (defaults: three X bays of 6 m and two Y bays of 6 m). Editing a bay moves every line beyond it.
3. Optionally switch on 45 degree diagonals, labels and bay dimensions.
4. To stop grid lines moving, use **Lock** and then **Grid** on the View tab.

See Section 10.5.2.

How do I place columns, beams and slabs?

1. Open the **Draw** tab and pick a tool.
2. **Column**: click to place. **Beam**: click the start and end, or type a length and press Enter. **Slab**: drag a rectangle. **Slab (poly)**: click the corners and close on the first point or press Enter.
3. Use **Select** to adjust; use **Erase** to delete what you click.
4. Edit sections, grades, spans and ends in the **Properties** dock.

Note: New columns line up with existing ones when **Square - 90 degrees only** is on. See Sections 10.6.1 and 10.3.

How do I choose the floor system and its loads?

1. Select a slab and open the **Properties** card.
2. Choose the slab system, for example RC solid slab on beams, RC flat slab, composite metal deck, ComFlor deck, precast hollowcore with topping or timber joisted floor.
3. Set the thickness, span direction, superimposed dead load and imposed-load category (default B1, office).

Note: The dead-load library already offers a partition allowance, so entering movable partitions as well double-counts. See Section 10.5.4.

How do I set the number of storeys and storey height?

1. Open the **Draw** tab and use the **Levels** bar.
2. Press **+** or **-** to add or remove storeys.
3. Name a level, give it its own height (1.8 to 20 m, default 3.5 m) or its own sheet.

4. Switch on the roof toggle to carry the roof as one extra level.

See Section 10.5.3.

How do I copy a floor to the level above?

1. Right-click the plan or 3D view and choose **Copy this floor to the level above**.
2. Pasting above the top level increases the storey count.

See Sections 10.5.3 and 10.6.4.

How do I add a footing under a column?

1. On the **Draw** tab, pick the **Footing** tool and click the column.
2. In **Properties**, choose the type: pad, strip, raft (local panel) or pile cap.
3. Check the allowable bearing (default 150 kN/m^2) and thickness (default 0.6 m).

Note: Footings are marked PF on the plan. A footing belongs to one column and is dropped if the column is deleted. See Section 10.6.6.

How do I draw a portal frame?

1. On the **Draw** tab, pick the **Portal** tool and draw the line of the frame span.
2. In the frame's card set eaves height, apex rise, spans, bay count and widths, pitch, bases (pinned or fixed), haunch and purlin options, and loads.
3. Type the net wind uplift and horizontal pressures yourself.
4. Press **Final design check** to run the full search; use the **Portal compare** view to see the quick and final designs side by side.

Note: Changing anything that affects the frame returns it to the quick design. See Section 10.6.5.

How do I start from a template building or portal frame?

1. In the **Templates** column on the left, open **Buildings** or **Portal frames**.
2. Choose, for example, Office (steel, composite deck), Office (RC flat slab) or Residential (RC flat slab), or a Single span, Two-span or Hit-and-miss portal shed.
3. To change bay sizes, edit the input boxes on the plan or 3D view; use **+** and **-** to add or remove a bay.

Note: Portal template loads are a placeholder, not a wind and snow assessment. See Section 10.7.

How do I get the programme to size the members to a target utilisation?

1. Open the **Tune** tab.
2. In **Sizing**, set the target utilisation (default 0.85, range 0.55 to 1.00) or per-element overrides.
3. Read the Green, Amber or Red band on each member in the plan, 3D and the **Results** view.

Note: A beam longer than the long-span flag (14 m) is flagged, not refused. See Sections 10.5.3 and 10.8.

How do I change the load factors?

1. Open the **Materials and factors** tab.
2. Edit gamma G (default 1.35) and gamma Q (default 1.50).

Note: Grades are set on the member, not here. See Section 10.4.

How do I set the lateral stability choice and the floor area?

1. Open the **Building** tab.
2. Choose the lateral stability option: Braced frame, RC shear walls or core (not modelled), or Moment frame (not checked).

3. Optionally type a **GIA override**; blank uses the drawn slab area times levels.

Note: Bracing and shear walls are placed and checked positionally, not designed. See Sections 10.5.3 and 10.8.

How do I see the model in 3D?

1. Under the View controls choose **3D**, or **Split** for plan and 3D side by side.
2. Drag the split ratio to suit.
3. On the View tab choose colour mode (scheme colours, utilisation, material, realistic or none) and projection.

See Sections 10.3 and 10.9.

How do I see column reactions and member diagrams?

1. On the View tab, use **Forces** to switch column reaction badges between the ULS design value and the SLS characteristic Gk + Qk.
2. Open the **Results** view for tables of slab panels, beams, columns, footings and portals.
3. Open the **Working** view for the audit sheet showing how each number was arrived at.
4. Use the diagrams for moment, shear and deflection per member.

See Section 10.9.

How do I undo and redo in Sketch-ET?

1. Press Ctrl+Z to undo and Ctrl+Y to redo, per change, in plan and 3D.
2. Alternatively use Undo and Redo on the **Edit** tab.

See Section 10.6.4.

How do I move, copy or rotate members precisely?

1. Open the **Edit** tab.
2. Use **Move by** (dX, dY, dZ), **Copy by** (count), **Move to** (X, Y, Z), the nudge arrows or **Rotate**.
3. Alternatively press M M to move, C C to copy or R R to rotate by pointing on screen.

Note: Type the two letters within a second. During a carried move or copy, X or Y restricts the direction and digits give a distance. See Section 10.6.4.

How do I control snapping while drawing?

1. Open the Snaps menu on the **View** tab.
2. Use the master switch and the individual snaps: Grid, Ends and centres, Midpoints, Drawing lines, **Square - 90 degrees only**, Angle 0/45/90 and Dimensions in line.
3. Turn off **Square - 90 degrees only** when tracing an irregular building.

See Sections 10.6.2 and 10.14.

How do I draw an elevation?

1. On the **Dimension** or **View** tab, pick the **Elevation** tool and draw a view line and box on the plan.
2. Double-click it to open the elevation.
3. Draw columns, beams, footings or portals in it; slab, comment and dimension tools are disabled there.

Note: Members land on the elevation's work plane, not on the view line. See Section 10.6.3.

How do I measure and dimension on the plan?

1. Open the **Dimension** tab.
2. Choose Linear, Aligned, Radius or Angle dimensions, **Area** or **Measure 3D**.

3. Use **Calibrate** to set the sheet scale from a known dimension.

See Sections 10.4 and 10.6.1.

How do I place the model on the OS grid and show a Google map behind it?

1. On the **Building** tab, open the **Site location (UK)** card.
2. Search for an address, postcode, place name, OS grid reference or `lat, lon`; tick the online lookup to use addresses and postcodes.
3. Type the **+Y** bearing (degrees clockwise from grid north) and, if needed, the **Level** in m AOD. Press **Refresh** to read the ground level again.
4. In the 3D view choose **Aerial**, **Street**, **Relief** or **3D** under Map in 3D. For the first three, set the ground drawn round the model: 250 m, 500 m (default), 1 km or 2 km.

Note: The map views are views; switch them off to edit. See Section 10.10.

How do I create a Sketch-ET report?

1. Open the **Export and IFC** tab.
2. Choose the paper size and orientation (default landscape), and tick **Plan image** and **3D image** if wanted.
3. Press **Word report (.doc)**, or **PDF report** and choose Save as PDF in the print dialogue.

Note: The Plan or Split view must be open for the plan image, and the 3D or Split view for the 3D image. See Section 10.12.

How do I export a Sketch-ET scheme as an IFC model?

1. Open the **Export and IFC** tab.
2. Press **Export IFC file**.

Note: It is a concept model of simple blocks at the serial sizes, not a fabrication model. See Section 10.12.

How do I send a Sketch-ET scheme to the IFC node for a carbon study?

1. Open the **Export and IFC** tab and choose **Send to the IFC node**. A second send updates the same node.
2. Press **Open the IFC node** to work on it.
3. Choose your carbon factors there, then return and press **Pull carbon back** so the SCORS band moves.

See Section 10.12.

How do I hand Sketch-ET column reactions to Column, Foundation or Base plate nodes?

1. Press **Run** on the Sketch-ET card first.
2. Wire the design's total output dot, or one of the per-column dots, to the Column, Foundation or Base plate node.
3. The characteristic base reaction arrives as a permanent and an imposed load in kN.

Note: Reactions are axial only, with no frame moments and no imposed-load reduction. See Section 10.13.

How do I upload an IFC model and see its quantities?

1. Add an **IFC viewer** node (tag "BIM · quantities").
2. Drop a plain IFC file on the node's drop area, or click it to browse.
3. Press **Open IFC module** and read the **Quantities** tab.
4. For a quick look beside the node map, press **Open IFC 3D viewer**.

Note: Compressed or XML files are refused with a message asking for a plain IFC export. See Section 11.2.

How do I load a second IFC model or replace the first?

1. To replace a file, press **Replace** on the node (or **Replace IFC** in the viewer toolbar).

2. To add another, press **+ Add IFC file** in the **Models loaded** list.
3. Tick the model the card should report on.

See Section 11.2.

How do I see the quantities for each storey?

1. In the IFC module, open the **Per storey** tab.
2. Read the floor area, slab count, volume and element total per storey.
3. Click a row to isolate that storey.

See Section 11.4.

How do I find a particular element in the model?

1. Open the **Elements** tab and search by name, material or type.
2. In the 3D viewer, right-click an element for its card with volume, areas, mass and carbon.

See Section 11.4.

How do I show only one storey or one category in the 3D viewer?

1. Use the storey selector on the viewer toolbar to show one storey at a time.
2. Use the category chips above the model to show or hide a category. Double-click a chip to isolate it, and press *Show all* to restore.

See Section 11.4.

How do I colour the model by carbon?

1. On the viewer toolbar, set **Heat map** to total $kgCO_2e$ or $kgCO_2e$ per m^3 per element.
2. Choose the scale: logarithmic, even spread by rank, or linear.
3. Use **Colour** to colour by element type or by material instead.

See Section 11.4.

How do I measure distances and areas in the IFC viewer?

1. For a distance, press **Measure** and pick two faces.
2. For an area, press **Area**, click points around a face and close the shape.

See Section 11.4.

How do I walk through the model?

1. Press **Walk** on the viewer toolbar.
2. Move with W, A, S, D; E and Q go up and down; Shift runs; Esc exits.

See Section 11.4.

How do I get the embodied carbon of a model?

1. Load the model and open the IFC module.
2. Read the **Quantities** tab and the **Working** tab for the carbon table.
3. Read the right panel: *Life cycle modules A1 to A5* and the SCORS rating.

Note: Only modules A1 to A5 are calculated. See Sections 11.4 and 11.13.

How do I choose which carbon factor is used for a material?

1. Open the **Charts** tab and click a bar to write that data point into the factor for the material.

2. To apply a pick to one element type only, use the **Applies to** control on the chart, or the *Carbon factors by element type* panel.
3. To pick for a single member, right-click it and use the **Factor:** selector.

Note: An element's own pick beats the element type pick, which beats the model-wide pick. See Section 11.5.

How do I see the lower, default and upper carbon totals?

1. On the **Charts** tab use the **Lower / Default / Upper** selector.
2. Read the three totals and the percentage spread in the right panel.

Note: Report the default and quote the range. With only custom factors there is no range. See Section 11.5.

How do I use a manufacturer's EPD in the carbon calculation?

1. Open the **Custom EPD factors** panel.
2. Store the reference file (PDF, XML or JSON), enter a name and a factor for concrete ($kgCO_2e/m^3$), reinforcement, structural steel or timber ($kgCO_2e/kg$).
3. Pick the new bar on its chart.

Note: A custom EPD publishes no range. See Sections 11.4 and 11.5.

How do I correct an element or category that is the wrong material?

1. For a whole category, open *Element classification and rebar* and set **Counted as** to concrete, steel, rebar, timber, masonry, aluminium, glass or other.
2. For one element, right-click it and change **Type**, **Material** or **Factor**.

Note: Element overrides survive a reload of the same file but are cleared when you load a different file. See Section 11.6.

How do I change the reinforcement allowance?

1. Open *Element classification and rebar*.
2. Type a node-wide allowance (default 110 kg/m^3), or use the thumb-rule picker.
3. Tick per-category thumb rules, or type a rate for a single category, which beats the thumb rules.

Note: If the model holds modelled reinforcement, every rate is set to 0. See Section 11.6.

How do I add quantities that are missing from the model?

1. Open the **Manual additions** panel.
2. Enter **Concrete m^3**, **Rebar kg**, **Steel kg**, **Timber kg** or **Masonry kg**.
3. They appear as a separate category called "Manual additions", which you can tick in or out.

See Section 11.8.

How do I include or exclude categories from the carbon totals?

1. In the right panel, use the *Element carbon split* list.
2. Tick or untick a whole type, or one type and material.

Note: Non-structural categories ("Other elements") start excluded. Re-loading a file never reverts your ticks. See Sections 11.3 and 11.9.

How do I set the transport and construction stages (A4 and A5)?

1. Open the *A4 transport and A5 construction* panel.
2. Choose the method: **IStructE per-material** (default) or **Percentage uplift (legacy)** (A4 12 percent, A5 4 percent, both editable).

3. Choose a transport scenario per material, or **Custom** and type a value.
4. To change any rate, press the lock button; press it again to restore the published values.

Note: Leave the A5.1 demolition area at 0 for a clear site. See Section 11.7.

How do I set the gross internal area and get a SCORS band?

1. Type the **GIA** on the SCORS card, or on the **Per storey** tab press **Use … m² as this model's GIA**.
2. Optionally type a **Custom m²** for a storey.
3. Read the band in the right panel.

Note: Changing GIA changes A5.2. Without a typed value the workspace figure is used. See Section 11.9.

How do I compare several IFC models?

1. Load at least two models.
2. Press **Compare models** on the viewer toolbar (up to five at once).
3. Read the metrics for each pane and the percentage spread under the panes.

Note: If models are not on the same bound and method, part of the difference is that setting. See Section 11.11.

How do I send a beam, slab, column or foundation from the model to a design node?

1. Right-click the member in the 3D viewer and choose **Design**, or use the **Design** tab.
2. Enter the area loads (gk and qk in kN/m²) and, for columns and foundations, the number of floors.
3. Wire the member's design dot into the matching beam, slab, column or foundation node.

Note: A design driven from the model is locked against other incoming wires. Remove the design link in the Design tab first to free it. See Section 11.12.

How do I send IFC quantities to a Math node?

1. Use the quantity output ports on the IFC node card: concrete volume, steel mass, rebar (est.), timber mass, total mass or A1-A5 CO2e.
2. Wire one to the **Math** node input.

Note: The totals honour the include/exclude ticks. See Section 11.10.

How do I produce the IFC report?

1. Open the IFC module and press the header **Report** button.
2. Keep the 3D viewer open if you want the 3D image captured.

See Section 11.14.

How do I add a Cut and Fill node and open it?

1. In the node library choose **Cut & Fill** (tag "Earthworks · DXF").
2. Press **Open cut & fill module** on the node card, or double-click the node.
3. To work with several surveys, use the header picker to switch, rename or press **+ Add cut & fill survey**.

See Section 12.2.

How do I import a survey into Cut and Fill?

1. Export an ASCII DXF in metres from CAD.
2. Drop it on the **Topo import (DXF)** panel, or click the panel to browse.

3. Read the import summary: entity counts, survey points, levelled points against points without a level, and rejected text.

Note: DWG files are refused. The module assumes metres. See Section 12.3.

How do I fix points that have no level?

1. In the import panel raise **Label match distance** slightly (default 1.0 m).
2. Re-check the contested points afterwards.
3. Alternatively, open the point review and type a level by hand, using **Show / hide levelless points** to list them.

See Sections 12.3, 12.5 and 12.13.

How do I review contested survey levels?

1. Open the point review and look at the contested points (shown first).
2. Type a level for a point, or exclude it. To handle them all at once use **Accept all nearest** or **Exclude all contested**.
3. Press **Sign off point review** when done.
4. To undo your changes press **Reset my overrides**.

Note: Sign-off does not change any number, and any later change un-signs it. See Section 12.5.

How do I leave trees or services out of the ground model?

1. Open the **DXF layers** panel.
2. Switch off the layer.

Note: The ground model and all volumes are recomputed. See Section 12.4.

How do I find mistyped spot heights?

1. Run the **Spike check** in the surface settings.
2. Adjust sensitivity (default 6) or "Ignore deviations below" (default 0.15 m) if needed.
3. Exclude flagged points singly or all at once.

Note: Nothing is excluded automatically, and the check cannot tell a typo from a retaining wall or kerb. See Section 12.6.

How do I remove long thin triangles over unsurveyed ground?

1. In the **Surface display** panel set **Drop triangles with a side longer than** to a value above zero.
2. Or trace a **Ground-model extent (clip outline)** around the surveyed area.

See Section 12.6.

How do I change the contours and the look of the surface?

1. In the **Surface display** panel keep **Generate contours** on.
2. Set the minor interval (default 0.25 m) and major interval (default 1.0 m).
3. Choose what to show in 3D and on plan, hillshade and level values.
4. Choose the views: **3D surface**, **Plan & review** or **Split**.

See Section 12.7.

How do I draw an excavation?

1. Pick a shape: Rectangle, Circle, Polygon, Trapezoid, Swale, Free form or Pond.
2. Set its parameters, centre (E, N), rotation and name, or drag corners on the plan.

3. For Free form, click the corners on the plan. For Pond, pick closed contours from the DXF plan, each at its own level.

Note: Rotation snaps to 15 degrees unless Shift is held. See Section 12.8.

How do I set the formation level?

1. By depth (default 1.0 m): the formation is that depth below existing ground at the shape's centre.
2. By level: untick "Set the formation by depth below existing ground" and type a formation level in metres on the survey datum.
3. Add a side batter (H:1V), base fall or build-up layers if needed.

Note: A blank formation level means no result, not zero. See Section 12.8.

How do I type a building footprint instead of drawing one?

1. Open the **Building footprint** panel.
2. Type at least three corner coordinates in true grid E/N, or draw on the plan.
3. Check the plan area shown and any warning about corners outside the survey.

See Section 12.8.

How do I dig for an IFC building on the survey?

1. Open the **IFC model** panel and load the model.
2. Enter the insertion point (E, N), rotation and base level. **Sit the base on existing ground** reads the ground level at the insertion point.
3. Choose **Dig for every buried element**.
4. If a row says "overlaps not fully separated", press **Work harder on the overlaps**.

Note: Such a row is over-stated. See Section 12.8.

How do I read the cut, fill and import or export volume?

1. Read the header chip: IMPORT x m^3 or EXPORT x m^3 (net fill minus cut), or BALANCED.
2. Read the node card for cut, fill and net across every solved excavation.
3. Open the results panel for plan areas, maximum and mean depths and ground model coverage.

Note: Volumes are in-situ (bank) volumes; apply bulking and compaction yourself. See Sections 12.9 and 12.10.

How do I draw a section through the ground and excavation?

1. Draw a section line on the plan.
2. Open the **Section view** tab in the bottom drawer, or the stage view.
3. Read the ground line, the shapes crossed and the cut and fill end-areas in m^2.

Note: End-areas must not be used to derive a volume. See Section 12.9.

How do I export the ground model to a BIM tool?

1. In the Cut & Fill module, use the ground model export to produce an IFC4 triangulated face set.
2. Open it in your BIM tool; the site carries the true-grid origin so it sits where the survey sits.

See Section 12.7.

How do I undo an edit or start again with a corrected DXF?

1. Use undo and redo for shapes, footprint, formation level and sections (40 steps). Loading a new DXF is not undoable.
2. To replace the survey, press **Delete the ground model** (two clicks to confirm); your shapes, sections and formation levels are kept.

3. Drop the corrected DXF in.

See Sections 12.2 and 12.7.

How do I produce the Cut and Fill report?

1. Open the module and press the header **Report** button.
2. Check the Validation section for any OK, CHECK or FAIL rows.

Note: The report prints the summed site total, not just the selected shape, and has no pass/fail banner. See Section 12.11.

A.5 SI map

How do I add a site investigation report to the map?

1. Add an **SI map** node from the node library and press **Open SI map**.
2. Drop a PDF or AGS file on **Add SI reports**, or click it to choose files.
3. Wait for the progress row to show the holes and bearing values found.
4. Press **Fit data** above the map to zoom to the report.

See Section 13.3.

How do I check or correct what the SI map read from a report?

1. Press **Review** beside the report in the **Reports** list.
2. Compare each value with the source page shown on the right.
3. Edit anything that is wrong; the map updates at once. Use **+ bearing value** to add a value by hand.
4. Tick **Reviewed** when you have checked the report.

See Section 13.5.

How do I put a report on the map when no location was found?

1. Open the report on the **Review** tab.
2. Press **+ Place site on map** and click the site on the map, or use **Look up postcode**.

See Section 13.5.

How do I find boreholes near my site?

1. Open the SI map and click the site on the map, or type a reference and press **Go**.
2. Set the **search radius** on the *Nearby boreholes* card.
3. Export the list with the CSV desk-study table or the Desk-study report (PDF).

See Section 13.4.

How do I show only clay at 1.5 m depth?

1. Open **Filters** and set the founding depth to 1.5.
2. Tick **Clay** in the soil types.
3. Set *Colour the holes by* to **Soil at the founding depth (Filters)**.

See Section 13.4.

How do I keep a copy of my SI library or move it to another computer?

1. Open **Library & privacy**.
2. Press **Export library (.json)** and save the file.

3. On the other computer, press **Import library** and choose the file.

See Section 13.8.

APPENDIX B

Troubleshooting

If something does not behave as you expect, find the symptom below. Every entry comes from the limits and cautions described in the chapters, and points you to the fix.

B.1 Workspace, node map, projects and reports

Symptom	Likely cause	What to do
A page shows "Civil ET failed to start"	A script threw during start-up, most likely because saved data from an older build does not suit the current one	Click Download backup, then Clear saved state and reload if needed. Send the backup and the error text to the support addresses on the About card.
My work has disappeared	Work is stored per browser and per site address; clearing site data deletes it, and it is not shared between machines	Open your saved project file with File > Open project.... Save project files regularly and keep them in your document system.
The wire is refused with "Invalid link for this node pair"	The pair is not on the allowed list (for example, a slab cannot feed a foundation directly)	Wire an allowed pair instead; check the table of which wires are allowed. Sticky notes never take wires and a node cannot connect to itself.
A wire is refused when I connect to a Math node ("delink it first (one wire per input)")	Each Math operand accepts one wire only	Delink the existing wire on that operand first, or use the other operand.
A wire into a slab, column, foundation or base plate is refused	The target cannot use the unit being carried (a slab accepts area loads only; column, foundation and base plate accept point loads only)	Wire a source that carries the unit the target accepts, for example an area load from a load set into a slab.
A new wire does not carry any data	Auto sync is OFF, or the wire's plug is paused	Turn Auto sync ON, switch the plug on, or press the card's ↻ button to push now.
I cannot edit a load or value in a design's input panel	Data arriving by wire is locked because every sync would rewrite it (the Node link chip says where it comes from)	Unplug or delete the wire, then edit the value by hand.
A design I dropped a wire on now says it is designed from another card	One source per design: the older wire was removed	Re-wire the source you want; the status line states which card now drives the design.
I dropped a second beam or slab card but no second design exists	Every card of a kind is a view of one workspace design list	Use the + Add button on the card to create a new design.
A second IFC model did not get its own card	Only the first IFC card is drawn; further models are folded into its model list	Look in the first IFC card's model list.
Dropping a file on the map shows "Only .ifc files can be dropped on the node map"	Any file other than an .ifc is swallowed with a warning so the browser never leaves the workspace	Drop an .ifc file, or open project files through File > Open project....
The Foundation, Column or Base plate card has vanished	Deleting a core card hides it rather than destroying it	Drag a fresh copy back from the node library.
Undo does nothing, or says "Nothing to undo"	History is per design, held in memory and lost on reload; undo is ignored while a dialog is open; Cut & Fill and Sketch-ET use their own undo	Undo from the module whose design you changed, close any dialog, and use the drawing-level undo in Cut & Fill and Sketch-ET.

Symptom	Likely cause	What to do
The page reloaded after I chose Open project, a stage or the example workspace	Modules read their state only at start-up, so these actions end with a reload	Expected behaviour. Save your project to a file before using them.
Changes since my last save are gone after opening a project	Opening replaces the workspace rather than merging	Save before Open, Open the example workspace, a stage switch or New project.
Save does not write to a disk file	Only Chrome and Edge write back to the same file; other browsers store a copy in browser storage	Use Save as... (.json) to download the project.
An old project file loaded only the foundation data	Files from before the workspace was included loaded the foundation data only and leave the node map as it is	Re-save from a current build once opened.
New project did not empty the map	New project resets the foundation side only	Delete or clear the node map separately.
A sticky note picture is refused	Browser storage is full, a note holds up to 6 pictures, and the project shares a budget of about 2.6 million characters	Remove pictures from other notes, or free browser storage.
A large IFC model reopens as bounding boxes	When storage is nearly full the model's 3D solids are kept for the session only	Load the IFC file again.
The map feels slow	A large workspace, or heavy cards (IFC, wind, Sketch-ET) holding geometry in memory	Collapse unused cards with ▾, zoom out to the tile level (below 30 %), delete heavy cards when finished, or split the job into RIBA stages or files.
The Report button is disabled in the foundation designer	A calculation is pending or has failed	Wait for the calculation to settle, or correct the failed inputs.
An alert says the report cannot be generated	The module's report could not be built from its current inputs	Check the module inputs and try again.
A figure is missing from the report	The corresponding view could not be drawn; for example, Sketch-ET plan and 3D captures use the on-screen view	Open the module's view first and regenerate; if the foundation report shows fewer figures than expected, open its views first and press Regenerate.
The PDF does not open or print	The browser refused to print a frame, or pop-ups are blocked	The report opens in a new tab instead; allow pop-ups. Choose Save as PDF in the print dialogue.
A foundation note says "plain concrete" and prints no bars	Reinforcement design is off for that foundation (a new pad starts with it off)	Turn reinforcement design on for that foundation and regenerate.
A downloaded report shows old numbers	A downloaded file is a snapshot taken when exported	Press Regenerate and export again.
The combined report's carbon for a design differs from the node map panel	The closing table uses the A1 to A5 figure printed in that design's own chapter; the row says so when they differ	Treat the row note as the explanation; for wind and cut and fill, the node-map carbon inventory stands in.
The combined report has no carbon donut chart	A donut is drawn only when no design has negative carbon (a timber sequestration factor can make a total negative)	Read the carbon table instead.
The combined report says "No report could be generated for the selected designs"	None of the selected designs could be generated	Check each design's inputs and try again.
Raft foundation cannot be selected	Raft appears in the type list but is locked ("in development, locked") in this release	Use pad, combined, strip or pile cap foundations.

Symptom	Likely cause	What to do
Timber beam gives no member design	Timber is analysis and take-off only in this release	Use concrete or steel for member design checks.
Carbon totals show 1000 m² as the GIA	With no typed value and nothing to derive from, 1000 m² is used and the panel says so	Type the real gross internal area in the right panel.
Two browser tabs seem to overwrite each other	Tabs on the same site share one local storage	Work in one tab at a time.
Wires between separate project files do not exist	Wires run between cards of the same map only	Keep linked designs in the same project.

B.2 Foundations, columns and base plates

Symptom	Likely cause	What to do
The Report button is greyed out	A calculation is pending or has failed; an open report closes in that case	Fix the input that causes the failure, wait for the calculation to settle, then press Report
Pad shows a thickness check instead of punching and crack checks	New pads start with Reinforcement design off and are designed as plain concrete to EC2 12.9.3	Switch **Reinforcement design** on in the Materials card
Load boxes are disabled with a note	A live link from a Column node is driving that load group	Delink the wire, switch off that transfer field, or turn off **Use column node** to edit by hand
Combined or strip footing does not receive column data	Incoming data is enabled only for pad foundations and pile caps	Enter column loads and sizes manually
Bearing check seems to ignore soil strength	The tool compares pressure with your SBC and does not compute bearing resistance from c, phi and unit weight	Take SBC from the site investigation and enter it yourself
Settlement is high but the verdict is not failing	Settlement is informational against a 25 mm indicator and is not part of the pass/fail verdict	Judge the settlement against the project limits yourself
Uplift or eccentricity check fails when the load is upward or very eccentric	The resultant lies outside the base or there is no net vertical load, so the base is not in contact	Increase the base size or weight, reduce the eccentricity, or review the loads; add top steel if hogging or lift-off is expected
Sliding utilisation shows 999	A horizontal action with no design vertical load leaves no friction to mobilise	Check the vertical loads and whether favourable loads are switched out
Factor of safety figures differ from the verdict	The 1.5 sliding and 2.0 overturning factors are shown for comparison only and are not EC7 verifications	Use the utilisation rows for pass/fail
Cover shortfall shows "review"	A user-defined cover is below the BS 8500 minimum	Raise the cover, or switch the cover mode back to auto
A bar size, spacing or fyk is named as substituted and the design fails	A cleared or invalid box was substituted	Enter a valid value in that box
Design fails on code scope	The concrete class is outside C12/15 to C50/60, the column lies outside the footing, interface friction is outside 0 to phi, fyk is outside 400 to 600 MPa, or similar	Correct the stated input; the reasons appear in the results panel and report

Symptom	Likely cause	What to do
Review note about wall-like shear	The column aspect ratio is over 2	Check the punching perimeter by hand; the truncated perimeter may overstate resisting length
Alternatives card is empty or out of date	The search runs only on its button and results clear when inputs other than size, depth, bars or column positions change	Press **Find reinforced alternatives** or **Find mass concrete alternatives** again
Alternatives search is slow on a combined footing	The search can take 5 to 15 seconds	Wait for it to finish
NHBC tab is disabled	The NHBC check does not apply to pile caps	Check tree influence separately for pile caps
NHBC check fails	The pad thickness plus depth of soil is less than the required founding depth	Increase the depth or thickness
Soil report values did not appear in the design	The tool does not interpret the PDF; the values must be typed in the "from report" boxes and applied	Type the values and apply them
"Updating design..." shows	The design's numbers have changed, so it is being recalculated	Wait for it to finish; unchanged designs are not recalculated
Column check fails on Class 4 scope	Class 4 CHS, EHS and back-to-back angles are outside the tool's scope	Choose a section that is not class 4, or design it outside the tool
Steel column tension capacity looks high	Net-section tension is not checked; only the gross plastic resistance is used	Check the net section by hand
Characteristic and factored loads mixed up on a column	The "The row values are" selector does not match what was typed; mixing is the most common error	Set the selector to match your numbers and re-check any rows typed before wiring
Load rows typed before a wire was connected give odd results	Rows the wire does not write are read as characteristic and factored with the wired rows	Re-check anything you typed before connecting
A wired load row went back to zero	The wire stopped driving that row, so it was reset to avoid stale loads	Re-enter the load or restore the wire setting
Wind loads do not appear on the column from a wind designer	The map allows the wire, but the column does not read wind rows from it	Enter wind rows by hand
Steel column ratio looks unconservative at the ends	Top moment left at 0, so the column is treated as having a single-ended moment	Enter both the bottom and top moments
RC column wind case looks sensitive	When wind relieves the axial load the lower axial pairing governs	Check the 1.0G sets and confirm the result in your own review
RC fire row shows "CHECK B" and fails	Method A validity limits are exceeded (for example effective length over 3 m, eccentricity over 0.15h, or reinforcement over 4 %)	Use a larger column, more axis distance, justify a lower fire load ratio, or check by Method B or Annex C outside the tool
RC column shows "more than the section can carry"	Required steel area is beyond about 64 times the provided area	Increase the section size or concrete class, or reduce the loads
RC cover or concrete class fails	Cover is below the bond or fire minimum, or the class is below the BS 8500 minimum for the exposure	Raise the cover or concrete class, or choose a milder exposure if justified
RC detailing fails	Bar size, bar count, link size, link spacing, area limits or bar gaps fall outside the clause limits (for example minimum bar 12 mm and at least 4 bars)	Adjust bars and links on the section view

Symptom	Likely cause	What to do
Column self-weight seems missing from the column check	It is not added to the column's own check, only to what it sends to the foundation and base plate	No action needed; check the column with the self-weight included in your loads if you want it
Base plate result fails with more than three anchors per side	ETAG design is limited to three per side and forces utilisation to at least 1.01	Use 3 or fewer per side, or switch ETAG design off
Base plate NEd shows zero	Negative NEd is treated as zero	Enter compression as a positive value
Base plate tension rows do not show	The anchors are not in tension	No action needed; they appear when the tension side is active
Base plate shear check is stricter than expected	Friction uses only the column compression; the anchors carry the rest	Switch off friction or set the coefficient to 0 only if the anchors are meant to carry all shear
Base plate section cannot be changed	The section is taken from the linked Column node	Change it on the Column, or turn off **Use column node data**
"Benchmark comparison mismatch" notice appears	The inputs match a built-in benchmark case such as 350 kN, 75 kN and a 600 x 600 x 40 plate	Ignore it for real designs
Splitting resistance has no effect in the anchor check	Splitting data can be entered, but the calculation treats splitting as always satisfied	Check splitting against the European Technical Assessment yourself
Weld result differs from your hand check	The weld factor and weld lengths are fixed standard values; stiffeners and shear keys are not modelled	Check any local detailing outside the tool
Concrete strength under the base plate cannot be edited	Alpha cc of 0.85 and gamma c of 1.5 are fixed in the node	Accept the fixed values

B.3 Beams, slabs, loads and steel sections

Symptom	Likely cause	What to do
Message "Beam is a mechanism - add another support to restrain rotation"	Fewer than two supports at different positions and no fixed support	Add a support, or make one support fixed
A new beam shows PASS with governing "Nominal cover" at about 90 %	Auto cover mode takes the computed minimum, so cover passes by construction	Treat it as a starting point and check the real checks, such as flexure and shear
Beam diagrams look smaller than the design moment	**Analysis read at** is SLS by default	Switch to ULS to read design actions; strength checks already use ULS
Non-linear or Vibration appear on the Analysis card but are greyed	They are not implemented ("separate future check")	Do not rely on them; use a separate method
Short deep beam deflects more than a hand calculation expects	Shear flexibility ignored with Timoshenko off	Tick **Timoshenko / shear** (effect is 10 % to 25 % on short deep members)
Fixed-fixed beam with unsymmetric load keeps simple end moments with Timoshenko on	Fixed-end load vectors stay the Euler-Bernoulli ones	Be aware of the limitation for fully built-in members
Concrete beam fails with compression steel (As2)	K above K', so steel is needed on the opposite face and fewer bars were provided	Add bars on that face, deepen the section or raise the grade
Beam fails 'Shear strut'	Shear exceeds the capacity even at 45 degrees	Increase the section or concrete class
Shear looks conservative near supports	Peak shear is used unreduced; load near the support is not credited	Accept as conservative, or increase the section if it fails

Symptom	Likely cause	What to do
Top face of a simply supported beam shows reinforcement demand	Monolithic end supports is on (hogging of 0.25 times the span moment)	Switch off if the beam rests on bearings, steelwork or masonry
Nominal cover shows "Review required"	Bond cover is met but durability cover or concrete class for the exposure class is not	Check the exposure class (the designer's decision) and raise cover or grade
Nominal cover shows FAIL	Bond minimum or casting-against-ground cover not met (40 mm blinding, 75 mm soil)	Increase cover
Fire check missing	Fire period set to 0	Choose 30, 60, 90, 120, 180 or 240 minutes
Crack check fails on a top face over supports	7.3.3 is checked on every tension face including top	Reduce bar size or spacing, or add steel
Concrete beam long-term deflection differs from slab for similar inputs	Beam creep coefficient is fixed at 2.0; the slab derives it from humidity and age	Remember the difference when comparing
Steel beam fails "Section class 4 (slender)"	Class 4 is flagged and not designed	Choose a stockier section
Steel beam fails web shear buckling	Web slenderness exceeds the limit and shear buckling to EN 1993-1-5 section 5 is not designed	Choose another section
LTB check unexpectedly governs	Restraint set to supports only or custom positions; a contraflexure point or fixed support is not a brace	Add real restraints, set **Fully restrained** if justified, or check C1
Overhang tip LTB looks odd	Unrestrained overhang uses the physical length with C1 = 1.0	Check the torsional restraint at the support yourself
Web bearing failure is reported but the beam still passes	Default "connection" setting reports but does not count web bearing	Set to "bearing on the bottom flange" if the beam bears on a flange
Warning that analysis E or I differs from the section	Custom inertia, or E not 210 GPa	Reset to the section values
Steel beam fails nothing for biaxial bending or torsion	These cases are out of scope and flagged	Check them separately
Timber beam shows only a deflection utilisation	Timber is analysis only	Design the timber member elsewhere
Quantities and carbon show zero, "not assessed"	The EC2 or EC3 design switch is off	Switch the design on
Reinforcement take-off reads low against a real schedule	The take-off uses straight bars, links and hooks	Use the bar-bending-schedule override (Rate or Total kg)
A wired beam load cannot be edited	Wired bands are locked	Use **Type my own loads instead**, or delete the wire
Wire rejected with a reason	Wrong unit sent (for example kN/m into a slab)	Fix the unit; the slab takes kN/m² only
Wind or Sketch-ET wire to a beam or slab brings no loads	The connection is permitted but loads are not written from those sources	Do not expect a transfer; enter loads another way
Loads node shows "UNITS ERROR - mixed"	The sigma total includes rows with different units	Untick sigma on mismatched rows
Row error "enter L x B or a direct area" or "enter the loaded width"	A required dimension is missing	Fill the missing dimension

Symptom	Likely cause	What to do
Downstream numbers look stale	Only the active load set tab is sent	Check which tab is active; check Auto sync or press Sync now
Imposed load is factored at 1.35	The row's class is G instead of Q	Correct the class
Math result is blank or an error	Division by zero, a mismatched unit on + or -, or a cycle	Fix the operands; remember there is no conversion between kN and N
Math refuses to add a number to a load wire	Adding, subtracting, max or min of a plain number against a multi-component load has no meaning	Use × or / for scaling, or wire a single-valued answer
Math from Steel sections gives 0	The node's data are not a number	Do not use a section in arithmetic
Section on a Steel sections node does not reach the column	Auto sync is off and no sync happened, or the wire was not drawn to that column	Press Sync now, or wire to the specific column input
Slab type list shows Flat slab, Ribbed slab or ComFlor greyed	They are locked in this release	Use one-way or two-way; a saved design with these opens as one-way
Default slab fails	l/d is about 1.4 times its limit, long-term deflection about 1.7 times span/250	Increase depth or bars
Slab warns that ly/lx exceeds 2	Coefficients are clamped at 2	Treat the panel as one-way
Failed "Load arrangement - coefficient validity"	Code coefficients used outside the option (c) conditions	Turn on the plate FE, or meet the conditions
Slab location fails with a doubly reinforced requirement	K above K' (shown as 1.25) and doubly reinforced slabs are not designed	Increase depth or grade
Slab shear fails	Shear exceeds the no-links resistance	Increase depth or steel; no shear reinforcement is designed
Punching fails under a point load	Bearing area, depth or steel too small; no links are designed	Enlarge the loaded area, deepen the slab or add steel
Smoothing warning on a slab	A concentrated load is driving the panel and smoothing removed over 15 % of the peak	Check the band width and set it by hand if needed
Slab FE is slow	High mesh density; work grows steeply with nodes	Use Coarse for dragging; Normal for design
Slab Loads wire rejected "kN/m"	Slab accepts only kN/m² area loads	Use **area** rows, or a Math result in kN/m²
Two wires drive the same slab field	The last synced wins	Pick distinct fields
Cover check is never the governing check on a slab	Automatic cover is set to the minimum so it passes by construction	Use user-defined cover if you want to test
Slab OUT load is lower than expected	Muted components are excluded downstream	Unmute them

B.4 Wind, Sketch-ET, IFC and Cut & Fill

Symptom	Likely cause	What to do
Wind results show "OUT OF BOUNDS"	The site is outside the area covered by NA Figure NA.1	Do not use the numbers; the node designs to the UK Annex only. Move the site to a UK location.
Warning that the marker is off the traced coastline	The site position is off the coastline traced from Figure NA.1	Check the Easting, Northing and search result; move the site or correct the grid reference.

Symptom	Likely cause	What to do
Site will not move when you click the wind map	The site is locked (a site found by search is locked)	Press **Unlock** in the Site and wind climate card.
Postcode or address search finds nothing	The online lookup is not ticked	Tick **Look up addresses, postcodes and ground level online**, or use a place name, grid reference or `lat, lon`.
Altitude not filled in	The online lookup is off	Tick the online option or type the altitude.
Basic wind speed warns it is defaulted	Manual mode with no vb,map typed	Type **vb,map** or switch to **From the map (Figure NA.1)**.
Qp changed after changing shore settings	The fetch to the shore changed, which changes ce(z)	Re-read the peak velocity pressure before relying on earlier numbers.
A narrow tidal river is treated as the shore	The water is wider than the ignore setting	Raise **Ignore tidal water narrower than** (default 0.5 km).
Shore distance wrong in Northern Ireland or for water the data misses	The built-in shore data covers Great Britain only	Use **Shore no further than** as a manual cap, or enter the distance yourself.
Fetch labelled "over 100 km (Fig. NA.7 limit)"	The chart is read only to 100 km	Expected; the chart value at 100 km is used.
Warning about a polygon plan	Code tables are for rectangular plans, so the enclosing rectangle is used	Review the result as an approximation with engineering judgement.
Warning that the pitch is under 5 degrees	Pitches under 5 degrees use the flat-roof table and are not interpolated	Check the roof type and pitch.
Warning that cs cd is not justified	The automatic mode uses 1.0 where the code does not clearly allow it	Use the table mode, Annex B or enter cs cd directly.
Warning that cd is not set	Table mode chosen without a cd value	Read cd from Figure NA.9 and type it.
Warning about the funnelling gap	Funnelling applies only where the gap to the facing building is between e/4 and e	Confirm the gap or untick the wall.
Funnelling tick has no effect in a wind case	The ticked wall is windward or leeward in that case	Expected; funnelling changes only side walls.
Warning that the fundamental frequency is assumed	Annex B used with a blank frequency (46/h, meant for multi-storey buildings over 50 m)	Type the actual fundamental frequency.
Warning for a building over 200 m	Charts are clamped at 200 m	Treat the numbers with caution.
Wind force differs from a hand calculation	Shortest fetch within the direction sector, directional factor over a 45 degree sector, h/d per direction, or rounded map speed	Compare each step on the Results tab (Section 9.9).
IFC import into the Wind designer refused	Only plain IFC files are accepted	Export a plain IFC file; compressed, XML and zip files are refused.
Cannot edit footprint after importing an IFC model into Wind	Footprint and height are locked	Press **Release the IFC lock**.
3D and flow tab says the model has not loaded	The 3D library did not load	Use the **Plan - zones** tab, which gives the same numbers.
Wind load seems to have the wrong sign in the receiving node	The wind node passes magnitudes only	The receiving node applies its own sign convention; apply combinations there.
Sketch-ET card sends nothing on its wires, says "not run yet"	The design has not been run	Press **Run** or **Open Sketch-ET**.
PDF underlay will not display	The PDF drawing library has not loaded	Export the plan as PNG or DXF instead.

Symptom	Likely cause	What to do
Sketch-ET scale looks wrong	Underlay not calibrated	Use **Calibrate** with a known dimension and check a second dimension.
Drawing will not land where you click	Snaps, notably **Square - 90 degrees only**, are pulling the pointer	Turn off the Square snap or the master snap.
Beam flagged as a long span	It is longer than the 14 m long-span flag	This is a prompt, not a limit; confirm by full design.
Warning that a braced bay does not reach the foundation	Bracing or shear wall stops below the top or foundation	Extend it to the foundation, or check lateral stability separately.
Sketch-ET sizes treated as final	The output is a scheme from thumb rules and tributary loads	Confirm every size by full design; there is no lateral, second-order, serviceability or connection design.
Portal frame sections change back to the quick design	Something affecting the frame changed after the Final design check	Press **Final design check** again.
Portal template uses the live design rather than the stored one	Target utilisation or partial factors differ from 0.85, 1.35, 1.50	Reset them, or run the Final design check.
Sketch-ET footing has no punching, bending or settlement check	Ordinary footings only get a bearing check	Design the footing in the Foundation node.
Large PDF underlay causes storage problems	The sheet image is stored with the design	Use smaller sheets.
Sketch-ET column reactions seem low for design	Reactions are axial only with no imposed-load reduction or moments	Treat as characteristic values for scheme use only.
IFC file refused or "No STEP entities found"	The file is compressed, XML, zip or not an IFC file	Export a plain IFC file.
Only boxes show in the IFC viewer after reopening	The solids were too large to store and the file has not been reloaded	Re-load the file.
Some elements are not shown in the IFC 3D preview	Solids are built for only the first 1,600 elements	The rest are still counted in quantities; a warning says so.
Cannot re-assign elements or design them in a large IFC model	Above 4,000 elements only category totals are kept	Re-load the file to work per element, or reduce the model.
Overrides disappear after loading another file	Overrides are keyed to the original file	Re-apply them; a warning says how many were cleared.
Carbon looks too low and "Other elements" are missing	Non-structural categories start ticked out	Tick them in the *Element carbon split* list if in scope.
Reinforcement carbon appears to be zero	The model contains modelled reinforcement, so the allowance is set to 0	Expected; reinforcement is not counted twice.
Steel or concrete counted in the wrong place	The material name was matched to a different bucket	Fix by category in *Element classification and rebar*, or by element in the right-click card.
SCORS band not shown on the IFC report	No GIA is set	Set a GIA before quoting a band.
Carbon range shows a single value	Only custom factors are in use, so there is no published range	Expected; use IStructE data points to see a range.
A5 changes when GIA changes	A5.2 is charged on GIA	Set the correct GIA first.
A5.1 adds carbon unexpectedly	A demolition area was entered	Leave A5.1 area at 0 for a clear site.
Cannot edit A4 and A5 rates	The panel is locked to published values	Press the lock button to customise.

Symptom	Likely cause	What to do
IFC Reactions tab is not usable	Marked Under development in this build	Do not use it for design.
Design wire from the IFC model is greyed or locked against other wires	A design driven from the model owns the load row	Remove the design link in the IFC Design tab first.
Design shows "not wired", "out of date" or as a stub	Not connected to a node, out of step with the model, or its element is no longer in the model	Wire it, refresh it, or remove the stub.
Report image of the 3D view is missing in the IFC report	The viewer was not open when the report was built	Open the viewer, then build the report.
Cut and Fill: "Only ASCII .dxf files are supported"	DWG or another format dropped	Export ASCII DXF (not DWG) from CAD.
Cut and Fill status NO LEVELS FOUND	Nothing in the file carried a level	Check the drawing has spot heights, 3D points, labels or contours with levels.
Cut and Fill status NO GROUND MODEL	Fewer than three levelled points	Add levelled points or fix the level matching.
Status shows DRAW AN EXCAVATION, SHAPE OFF THE SURVEY or NEEDS A FORMATION LEVEL	The first blocking item is missing	Draw a shape, bring it inside the survey, or set a formation level.
Cut and Fill volumes look wrong by orders of magnitude	The drawing is not in metres (the units setting is not used for scaling)	Convert the survey to metres before export.
Levels dropped as "outside the level band"	Text was not a decimal with two or three decimals, or was far from the rest of the drawing	Check the labels; whole numbers and long numbers are never taken as levels.
Many points show "no level"	Level text is too far from the points	Raise **Label match distance** slightly and re-check contested points.
Footprint outside the survey or coverage below 99.5 percent	Volumes outside the ground model are not counted	Keep the footprint inside the survey extent.
Total looks too large when shapes overlap	Overlapping shapes are each counted in full	Draw one outline per separate dig, or use one free-form shape.
Coordinate system setting does not change the numbers	It is only a label	True Easting and Northing are used as they are; nothing is transformed.
Contours missing across a gap	They are suppressed across long triangles	Adjust the suppression setting or drop long triangles.
Report Validation shows CHECK	Review not signed off, contested points remain or points have no level	Review and sign off the points.
Cut and Fill results differ between versions	The module is in alpha	Check results independently before relying on them.

B.5 SI map

Symptom	Likely cause	What to do
A report is read but is not on the map	No site location was found in the report	Open it on the Review tab and use **+ Place site on map**, or look up the postcode
A bearing value looks wrong	The reader mis-read the text, unit or footing size	Compare it with the source page on the Review tab and correct it

Symptom	Likely cause	What to do
The Google map is not shown	No Google map key is connected to this copy of Civil ET	Use the OS or Esri base map
BGS borehole records do not appear	The map is zoomed out too far	Zoom in to within about 12 km
The SI library is empty after clearing browser data	The library lives only in the browser profile	Export the library regularly and import it again

APPENDIX C

Glossary

Terms and abbreviations used in Civil ET and in this guide.

A

A- / A+ - Header buttons that scale the large display text from -6 to +2.

A0 - The product-stage carbon module that the IFC node takes as zero for buildings.

A1 to A3 - The product stage of upfront embodied carbon: raw materials, transport to the factory and manufacture.

A1-A5 - The life-cycle stages used for embodied carbon: product stages A1 to A3, transport to site (A4) and construction (A5).

A4 - The embodied carbon of transporting materials to site.

A5 - The embodied carbon of the construction process, made up of A5.1 to A5.4.

A5.1 - Pre-construction demolition carbon, charged on the demolished area.

A5.2 - Construction activity carbon, charged on the gross internal area.

A5.3 - Carbon from material waste on site.

A5.4 - Carbon from worker transport, entered as a value.

About card - The information panel opened by the i button, showing the version, a short description, the verify-your-results notice and contact addresses.

Action basis - The way load components are grouped when factoring: correlated by action (G, Q, W) or independent by component.

AGS - The standard file format for exchanging geotechnical and ground-investigation data.

Align tops - A right-click option that lines up the top edges of two or more selected cards.

Alpha - A very early, preliminary status carried by the Cut and Fill module; results must be checked independently.

Alternative designs - A list of other sizes, depths and bar arrangements that pass every check, ranked by lowest embodied carbon.

Altitude factor - The factor that increases the basic wind speed with the height of the site above sea level.

Analysis read at - The switch choosing whether diagrams and reactions are drawn at SLS or ULS.

Anchor - A bolt that holds a base plate to the concrete foundation.

Annex B - The detailed procedure in the wind standard for working out the structural factor.

Area load - A load per unit area in kN/m².

Auto cover - A cover mode that sets cover to the computed minimum.

Auto sync - The master switch for the whole map that makes wires carry data live; ON by default.

Axis distance - The distance from the concrete surface to the centre of the main bars, used in the fire check.

B

Bank volume - The in-situ volume of ground before bulking or compaction.

Bar bending schedule - A table of bar marks, shapes, diameters, lengths and numbers, with BS 8666 rounding.

Bar-bending-schedule override - A way of replacing the derived reinforcement quantity with your own rate or total.

Base plate - The steel plate welded to the foot of a steel column and bolted down to the foundation.

Basic wind speed - The wind speed taken from the map or typed in, before the altitude factor, written vb,map.

Batter - The slope of the sides of an excavation, given as horizontal to vertical.

Beam analysis - The node that analyses a continuous beam and designs it in concrete or steel.

Bearing pressure - The contact pressure between the underside of the base and the ground.

Beta (punching) - The factor that increases the shear force at the punching perimeter to allow for eccentric load or moment.

BGS - The British Geological Survey, whose public borehole records the SI map can show.

Blue Book - The Steel for Life publication of section properties and resistances whose data the app embeds.

Braced - A column in a frame where sway is prevented by other elements.

BS 8500 - The UK standard that sets the minimum cover and concrete class for an exposure class.

Buckling - Sideways instability of a slender member under compression.

Build-up - A set of named layers with thicknesses used to define an excavation.

Bulking - The increase in volume of excavated ground, which Cut and Fill does not apply.

C

Calc description / scope - A free-text box in the foundation report window whose text appears in the report header.

Calibrate - To set the drawing scale by clicking two points and typing the real distance.

Carbon panel - The right-hand workspace summary on the node map totalling quantities and embodied carbon.

Characteristic combination - A service combination of Gk plus Qk.

Characteristic loads - Unfactored loads as they are expected in service.

Class (G, Q, H) - The label of a load as permanent, variable or horizontal.

Class 4 section - A slender steel section where local buckling limits resistance; the app flags it but does not design it.

Clear links - The button that removes every wire on the map at once.

cmin,dur - The minimum cover needed for durability.

Combination 1 and 2 - The two sets of partial factors in Design Approach 1.

Combined footing - One base carrying two or more columns.

Combined report - The single document produced from the node map's Report button, with one chapter per chosen design.

Compaction - The reduction in volume of placed fill, which Cut and Fill does not apply.

Component method - The EN 1993-1-8 way of checking a joint by its separate parts, such as T-stubs, bolts and welds.

Contested point - A survey point whose level label has a rival label reading a different level nearby.

Continuity - Whether a slab edge or support carries on into an adjacent span.

Contour - A line joining points of equal ground level.

Core card - One of the three always-present cards (Foundation, Column, Base plate); deleting one hides it instead.

Correlated actions - Treating the vertical, horizontal and moment parts of one action (G, Q or W) with the same factor.

Correlation factor - The reduction applied to the sum of windward and leeward wall forces depending on the height to depth ratio.

Country terrain - A terrain setting that takes the town factor as 1.0, which is conservative.

Cover - The distance from the surface of the concrete to the nearest bar.

Crack width - The calculated width of flexural cracks under the quasi-permanent load, compared with your limit.

Cracked concrete - Concrete assumed to contain cracks in the anchorage zone, which gives lower anchor resistances.

Creep - The long-term increase of deformation under sustained load.

Creep coefficient - The factor that describes long-term deformation of concrete under sustained load.

Curtailment - Stopping some bars short of the support, where less steel is needed.

Curved / Straight - The two wire styles, bezier curves or right-angle bends.

Cut - Volume where the ground is above the formation level and must be removed.

D

Data route - The choice of table shown for a Blue Book section.

Deflection - The vertical movement of a member under load.

Delete connection - The wire menu option that removes that wire.

Demo - The amber header button that starts the guided tour.

Design Approach 1 - The Eurocode 7 method that checks two combinations of partial factors.

Design list - The single set of designs (for example B1, B2) shared by every card of one kind.

Design loads - Loads already multiplied by partial factors.

Design picker - The header drop-down for choosing between a module's designs.

Design working life - The period, 50 or 100 years, for which the structure is intended to be used.

Dimensions - A node holding simple geometry values (lengths, area) that can be wired onward to Math.

Direct-stiffness model - The finite-element method the beam uses to solve forces and movements.

Directional factor - The factor, cdir, that reduces wind speed according to the wind direction relative to the building.

Doubly reinforced - A section needing compression steel as well as tension steel.

E

EC2 (Eurocode 2) - EN 1992-1-1, the Eurocode for concrete design.

EC3 (Eurocode 3) - EN 1993-1-1, the Eurocode for steel design.

EC7 (Eurocode 7) - EN 1997-1, the Eurocode for geotechnical design.

Eccentricity - The offset of the load resultant from the centre of the base.

Effective length - The length of an equivalent pinned column used for the buckling check.

Effective span - The span used in analysis, measured between effective support points.

Element carbon split - The list in the IFC node where element types and materials can be ticked in or out of the carbon totals.

Elevation - In Sketch-ET, a view along a line drawn on the plan, used for roofs, sloping beams and portal frames.

Embodied carbon - The carbon emitted in producing, transporting and installing the materials of a design, totalled in kgCO_2e or tonnes of CO_2e.

EPD - An environmental product declaration, giving a manufacturer's own carbon factor for a product.

EQU - The overturning limit state, checking rigid-body rotation about a base edge.

ETAG 001 / TR029 - The European guidance used to check anchors in concrete.

Example workspace - The worked, wired demonstration map shown to a new browser and reopenable from the File menu.

Exposure class - The BS 8500 category of environmental conditions, such as XC2, which sets the cover and concrete class.

Exposure factor - The factor, ce(z), read from the National Annex chart using height and distance to the sea.

F

Fetch - The distance upwind to the shore, measured for each wind direction.

Fill - Volume where the ground is below the formation level and material must be added.

Final design check - The full portal frame design search in Sketch-ET that stores the chosen sections.

Fire load ratio - The ratio of the fire design load to the normal design load.

Fire period - The required fire resistance in minutes for a beam, or a REI rating for a slab.

Fit - The zoom button that fits every node in view without going above 100 %.

Fixed support - A support that restrains vertical movement and rotation.

Flexure - Bending resistance of a member.

Formation level - The finished level of an excavation floor, in metres on the survey datum.

Foundation node - The node on the map that lists the foundation designs and their utilisations.

Founding depth - The depth below ground level at which a foundation bears, used to read the soil on the SI map.

Free form - An excavation shape whose corners are clicked on the plan.

Friction coefficient - The factor that converts the column compression into shear resistance at the plate-to-grout interface.

Funnelling - Increased suction on side walls between neighbouring buildings, applied wall by wall.

G

GIA - Gross internal area, used to divide embodied carbon into an intensity and to charge A5.2.

Gk - The characteristic permanent action.

Governing horizontal force - The largest overall horizontal wind force over all directions.

Governing set - The load combination that gives the highest utilisation.

Green, Amber, Red - The bands Sketch-ET gives each member according to its utilisation against the target.

Ground model - The triangulated surface built from the surveyed levels.

Group - A named, coloured frame behind several cards that moves them together.

H

Heat layer - A colour wash on the SI map showing the bearing values your reports state, only where they state them.

Hillshade - A shaded rendering of the ground surface from a set sun direction.

Hogging - Bending that puts the top face in tension.

I

IFC - A BIM model file format; the IFC viewer node loads it and takes off quantities and embodied carbon.

Imposed load (qk) - The variable load from occupancy, in kN/m^2.

Imposed load category - The EN 1990 category of use that sets the combination factors.

Imposed-load category - The use category of a floor, such as B1 for offices, which sets the imposed load.

IN dot - The left-hand connection point on a card or design row where a wire arrives.

Interface friction - The friction between the base and the soil, expressed as a ratio of the soil friction angle.

Internal pressure coefficient - The coefficient, cpi, for pressure inside the building, which depends on openings.

K

K' - The limit of the bending parameter K above which compression steel is needed.

Kern - The middle-third zone of the base within which the whole base stays in contact with the soil.

L

Lateral restraint - A brace that stops a steel beam moving sideways or twisting.

Lateral-torsional buckling (LTB) - A buckling failure of an unrestrained steel beam in which it twists and moves sideways.

Layer - A named group of drawing entities in a DXF file that can be switched on or off.

Leave-one-out check - The SI map test that holds each reported location back, colours it from its neighbours and reports how far the colour can be from the report.

Level of detail - Below 30 % zoom, cards are drawn as simple tiles to keep a large map smooth.

Lift-off - Partial loss of contact between the base and the soil when the resultant lies outside the kern.

Link - A closed stirrup bar around the main column bars.

Live load set - A Loads data set whose values reach every downstream node as you type.

Load set - A tab of load rows in a Loads node.

Loaded width - The width over which an area load is turned into a line load.

Loads - A node holding load data sets of point, line and area loads by class.

Loads node - The node that holds load sets and passes their totals on by wire.

Look-up (online) - The optional tick that sends typed addresses, postcodes and site positions to online services.

Lower, default, upper bound - Three pricings of the whole model at the guide's low, recommended and high factor values.

M

Mass concrete - Concrete without reinforcement design, checked as plain concrete.

Math - A node with two operands (A and B) and an operator, used to build a load from dimensions and loads.

Math node - The node that applies one operation to two values while carrying units.

Mechanical anchor - An anchor that resists pull-out through its own mechanical expansion.

Mesh density - The number of plate divisions used in the slab finite-element solution.

Monolithic end support - An end support assumed to give some fixity, which creates hogging at a pinned end.

Mute - To keep a load on screen but leave it out of every calculation.

N

National Annex - The UK document that sets national values and procedures for a European standard.

Net bearing allowance - An option that adds the weight of removed soil to the SBC.

Net volume - Fill minus cut; positive means import and negative means export.

NHBC 4.2 - The NHBC Standards Chapter 4.2 guidance on building near trees and the required founding depth.

Node - A card on the map representing one design or item, such as a beam, slab or load set.

Node library - The searchable list on the left of the map from which nodes are added.

Node link chip - The header chip in a module saying how many wires feed it, which node supplies it, and whether Auto sync is off or the wire is unplugged.

Node map - The home screen of cards and wires on which a job is built.

Nominal cover - The specified cover, including the allowance for construction deviation.

Non-cracked concrete - Concrete assumed to be uncracked in the anchorage zone.

O

Option (c) - The UK National Annex arrangement that allows all spans loaded with 20 % redistribution.

Orography - The effect of hills and ridges on wind speed, entered as the orography factor.

OS grid reference - A UK Ordnance Survey National Grid position, held to the metre.

OUT dot - The right-hand connection point on a card or design row where a wire starts.

Overburden relief - The reduction in net pressure from the weight of soil that was removed.

P

Panel case - The combination of continuous and discontinuous edges of a two-way slab.

Peak velocity pressure - The gust pressure, qp, used to calculate wind pressures on the building.

Pile cap - A concrete block that spreads a column load into a group of piles.

Pin - The attachment that sticks a note to a card or group so it travels with it.

Pinned support - A support that restrains vertical movement but not rotation.

Plain concrete - Concrete designed without reinforcement.

Plasticity index - A measure of how plastic the clay is, used in the NHBC tree check.

Plate FE - The finite-element plate solution used for slab moments and deflection.

Plug - The control beside a wire's dot that pauses or resumes that wire's live sync.

Point load - A concentrated load at a single position.

Pond - An excavation shape made from a stack of closed design contours, each at its own level.

Portal frame - A single-storey frame drawn as a span line and analysed as a whole frame in Sketch-ET.

Preset - A library value that can fill a field in a Loads row or slab load.

Prismatic TIN method - The volume method that integrates prisms between the ground triangles and the formation.

Probability factor - The factor, cprob, that adjusts wind speed for the annual probability of exceedence.

Project bar - The strip across the top of the node map holding RIBA stage, job number, project name, designed by, reviewed by, client and date.

Project file - The single saved file that holds the whole workspace, project details and saved stages.

Project info and metadata - The File menu form for project name, number, designed by, reviewed by, client, date, RIBA stage and notes.

Pry-out - An anchor failure in shear where the concrete behind the anchor breaks away.

Psi factors - Combination factors that reduce variable actions in service combinations.

Pull-out - An anchor failure where the anchor is drawn out of the concrete.

Punching shear - A failure in which a column or pile pushes through a slab or base along a perimeter.

Q

Quantity override ticks - Tick boxes that include or exclude each element from both the carbon totals and the quantities.

Quasi-permanent combination - The load combination of permanent load plus a fraction of variable load, used for crack width.

Quick design - The live portal frame design that updates as you edit.

R

Raft - A foundation slab under a whole structure; locked in this version.

Rebar allowance - A thumb-rule mass of reinforcement per cubic metre of concrete added when none is modelled.

Reclassification - Correcting the type, material or factor of an element or category.

Redistribution ratio - The ratio of moments after redistribution to the elastic moments.

Regenerate - The report window button that re-reads the live design and redraws the report.

Report colour scheme - The choice of Amber engineering, Black / white, Corporate blue or Reading comfort for headings and table heads in a report.

RIBA stage - A stage of the RIBA Plan of Work, 0 to 7, used to stamp reports and to save snapshots of the design as it evolves.

RIBA-stage trend - A carbon-panel chart pricing past stages from the whole-map A1 to A5 split stored with each snapshot.

Roller support - A support equivalent to a pinned one in this planar model.

Roof zones - The named areas of a roof (for example F, G, H and I on flat roofs) with their own pressure coefficients.

S

Sagging - Bending that puts the bottom face in tension.

SBC - Safe (allowable) bearing capacity, the allowable ground pressure you take from the site investigation.

SCI P398 - The Steel Construction Institute guide whose simplified route is offered for base plate checks.

SCORS - The IStructE embodied-carbon rating scheme; the panel rates the project against its gross internal area.

Season factor - The factor, cseason, that reduces wind speed for temporary works in a known season.

Second order effects - The extra bending in a slender column caused by its own deflection.

Section - A profile line drawn through the ground and excavation, with cut and fill end-areas.

Self-weight - The member's own weight, added as a permanent load.

SI (site investigation) - The ground investigation reports for a site, such as boreholes, trial pits and laboratory results.

Sigma total - The sum of the ticked rows in a load set.

Sign off point review - A record that someone has looked at the contested survey points; it changes no numbers.

Simple report / Detailed report - The two foundation report modes, the first shorter than the second.

Sketch-ET - A node for sketching a scheme over a PDF or DXF, sizing elements at concept stage and handing reactions on to detailed nodes; labelled alpha.

Skin reinforcement - Side bars reported for deep beams.

Slenderness - The ratio of effective length to radius of gyration.

Sliding - The check that the horizontal action does not push the base along the ground.

SLS - Serviceability limit state, where deflection and cracking are checked.

Smoothing width - The band across which a concentrated-load peak is averaged in the slab.

Snap - A zoom-strip switch that lines dragged cards up with nearby cards and lands them on a 20 px grid.

Soil modulus (Es) - The stiffness of the soil, used for settlement.

Spike check - A test that flags survey points that deviate sharply from the plane through their neighbours.

Spot height - A surveyed ground level at a single point.

Stage snapshot - The saved copy of the whole design at a RIBA stage.

Status line - The message line by the Auto sync strip that reports what the map has just done, such as a refused wire.

Steel sections - A node giving section properties from the Blue Book to other nodes.

Steel sections node - The node holding a Blue Book section that can feed a column.

Sticky note - A node for typed notes, handwriting and pictures that carries no design data.

Stiff bearing length - The length over which a load bears on a flange, used for the web bearing check.

Strip footing - A continuous wall footing designed per metre run.

Structural factor - The product cs cd, which allows for size and dynamic response of the structure.

Strut - The compression member of the shear truss model in a concrete beam.

Strut-and-tie - A model in which concrete struts and steel ties carry the load, used for pile caps.

Surcharge - Extra load on the ground above the base.

Swale - A shallow channel excavation with side slopes along a path.

T

T-stub - An idealised part of a joint used to model plate bending in tension or bearing in compression.

Take-off - The quantities of concrete, rebar, structural steel and timber measured from the designs on the map.

Take-off report - A report with no pass/fail verdict, such as the IFC quantities report, marked TAKE-OFF.

Thumb rule - A rule of thumb, such as a span to depth ratio, used for concept sizing.

Timoshenko - An analysis option that adds shear deformation to a beam.

TIN - Triangulated irregular network, the ground surface made of triangles between surveyed points.

Top reinforcement - Bars near the top face of a base, for hogging or lift-off.

Town terrain - A terrain setting that applies the town correction factor from the National Annex.

U

UDL - Uniformly distributed load along a length.

ULS - Ultimate limit state, where strength is checked.

Underlay - A PDF, DXF or image placed behind the drawing to trace over.

Uplift - The check that the weight of the base and soil resists upward forces and water pressure.

Utilisation - The design effect divided by the resistance; at most 100 % passes and the app colours it green, amber (above 85 %) or red (above 100 %).

V

Velocity pressure zones - The wall zones A to E that carry different external pressure coefficients.

W

Weld throat - The effective thickness of a fillet weld, equal to the leg size divided by the square root of 2.

Wind designer - The node that works out wind actions on UK buildings to the Eurocode and its UK National Annex.

Wind load (WL) - The variable load from wind, entered as its own group.

Wire - A live link carrying data from the OUT side of one card to the IN side of another.

Wire label - The text on a wire naming the pair it links, for example "SLAB LOADS -> BEAM".

Wood and Armer moments - Slab design moments that include the effect of twisting moments.

Work plane - The plane on which members drawn in an elevation are placed.

Z

Zoom strip - The row of controls at the bottom of the map: -, readout, +, 100, Fit, Snap and wire style.

APPENDIX D

Index

Entries are section titles and how-to questions, in alphabetical order, with the page where each starts.

#

3D and flow 126
ψ factors and SLS combinations 77

A

A tour of the screen 6
A1 to A3: product stage factors 160
A4 and A5: transport and construction 162
A4 transport to site (Table 2.7) 162
A5 construction process 163
Action basis (Ch. 3) 29
How do I add a Beam node and open the full workspace 220
How do I add a Cut and Fill node and open it 245
How do I add a footing under a column 240
How do I add a Loads node and create a load set 227
How do I add a node to the node map 201
How do I add a parapet, curved eaves or mansard eaves to a flat roof 233
How do I add a point load, UDL or moment to a beam 221
How do I add a second beam, slab or foundation design 201
How do I add a site investigation report to the map 248
How do I add a Steel sections node and pick a section 226
How do I add a Wind designer node and open it 231
How do I add and use a sticky note 204
How do I add another foundation design 214
How do I add point loads and line loads to a slab 229
How do I add project details to reports 213
How do I add quantities that are missing from the model 244
How do I add several columns to one Column node 214
How do I add the diagonal wind directions 235
How do I add top reinforcement 211
Adding and using it 99
Adding it (Ch. 7) 96
Adding nodes (Ch. 2) 12
Adding reports (Ch. 13) 184
Adding the node and loading a model 156
Adding the node and opening the module 171
How do I allow for funnelling between buildings 235
How do I allow for orography 235
Alternative designs (Ch. 3) 32
Alternative-section searches (Ch. 4) 53
How do I analyse a continuous beam with several spans 220
How do I analyse a timber beam 225
Analysis (Ch. 8) 107
Analysis modes (Ch. 6) 76
Analysis settings (plate FE) 107
Anchor layout (Ch. 5) 65
Anchors and fastenings 198
Appearance (Ch. 16) 196
How do I attach a reference document to a Loads node 228
How do I attach a soil report 213

B

Base plate (Ch. 14) 191
Base plate details 65
Basic wind speed and peak velocity pressure 124
Beam (Ch. 14) 191
Beam (Ch. 16) 196
Beams, slabs, loads and steel sections (App. A) 220
Beams, slabs, loads and steel sections (App. B) 254
Bearing (EC7 6.5.2) 30
Bearing Capacity card 26
How do I bend or reshape a wire 202
Bending axis (Ch. 6) 76
Bottom drawer (Ch. 11) 160
How do I bring back a deleted Foundation, Column or Base plate card 204
How do I bring in an IFC model to give the building shape 234
Building (Ch. 10) 143
Building footprint (typed outline) 175

C

Calculation - what is checked 29
Calculation (Ch. 4) 51
Calculation basis (Ch. 10) 146
Calculation method (Ch. 11) 162
How do I cap the distance to the shore for Northern Ireland or missed water 235
Card controls (Ch. 2) 13
Categories and material buckets 157
How do I change the contours and the look of the surface 246
How do I change the load factors 240
How do I change the plan size and height of the building 233
How do I change the reinforcement allowance 244
How do I change the RIBA stage 205
How do I change the season factor or the return period 235
How do I change the span of a beam 221
How do I change the theme, font and text size 206
Chapters (Ch. 14) 190
Characteristic or design rows 47
Charts (Ch. 11) 161
Charts (Section 9.6.6) 127
How do I check anchorage, laps and curtailment on a concrete beam 224
How do I check crack control on a concrete beam 224
How do I check deflection of a concrete beam 224
How do I check fire resistance of a concrete beam 223
How do I check or correct what the SI map read from a report 248
How do I check pile reactions without designing the cap reinforcement 212
How do I check punching under a point load on a slab 230
How do I check shear in a concrete beam 223
How do I check the NHBC tree-influence founding depth 212
How do I check the wind node against my hand calculation 238
Checking against a hand calculation 128
Checks (Ch. 6) 82
Checks carried out 50
How do I choose a steel column section and grade 215
How do I choose between correlated and independent load actions 210
How do I choose the base plate calculation style 218
How do I choose the deflection limits for a steel beam 225
How do I choose the EN 1990 load combinations for a beam 222
How do I choose the floor system and its loads 239
How do I choose the foundation type (pad, combined, strip, pile cap) 208

How do I choose the internal pressure basis 236
How do I choose the panel case of a two-way slab 229
How do I choose the slab analysis basis and mesh density 230
How do I choose the slenderness and imperfection basis for an RC column 217
How do I choose the STR combination expression and punching options 210
How do I choose which carbon factor is used for a material 243
Choosing a calculation basis 64
Code coefficient tables 107
How do I colour the model by carbon 243
Column (Ch. 14) 191
Column and base plate 195
Column details (Ch. 5) 65
Column to Foundation wiring 33
Combinations (EN 1990) 105
Combined footing (CF) 25
Compare (Ch. 9) 127
How do I compare normal and funnelling pressures 237
How do I compare several IFC models 245
Comparing models (Ch. 11) 165
Concrete beam design (EC2) 78
Concrete outputs and governing utilisation 81
Concrete, geotechnical and loading guidance 199
How do I connect two nodes with a wire 201
Connections (Section 5.7) 69
Connections (Section 7.3.3) 102
Connections to other nodes (Section 10.13) 148
Connections to other nodes (Section 12.12) 177
Connections to other nodes (Section 4.8) 53
Connections to other nodes (Section 6.9) 85
Connections to other nodes (Section 8.9) 112
Connections to other nodes (Section 9.10) 128
Contribution to the workspace 164
How do I control snapping while drawing 241
How do I copy a floor to the level above 240
How do I correct an element or category that is the wrong material 244
Cover and status block 189
Cover, fire, anchorage and curtailment 80
Crack control (EC2 7.3.3) 110
Crack control (EC2 7.3) 80
How do I create a custom RIBA stage 206
How do I create a Sketch-ET report 242
Cut & Fill 192

D

Default data points 161
Deflection (EC2 7.4) 110
How do I delete a node 203
Deleting nodes - what else goes 16
Describing the building 124
How do I design a combined footing 211
How do I design a one-way or two-way slab 229
How do I design a pad foundation for a column load 209
How do I design a pile cap 212
How do I design a reinforced concrete beam to EC2 222
How do I design a reinforced concrete column 216
How do I design a signboard, free-standing wall, tower or mast instead of a building 236
How do I design a steel beam to EC3 224
How do I design a strip footing 212
How do I design anchors as ordinary bolts 219
Design Approach 1 29
Design checks (Ch. 8) 109
Design forces (Ch. 5) 64
Design options and long-term inputs 107
Design ports (Ch. 11) 165
Design standards (Eurocodes and British Standards) 198
How do I design the beam about its minor axis 221
How do I design the top steel over a pinned end of a monolithic beam 223
Designing from the model 165
Detailing, cover, fire, anchorage 110
How do I dig for an IFC building on the survey 247
Dimensions node (Ch. 7) 101
Directional factor (Ch. 9) 125
Disclaimers (Ch. 14) 190
Distance to the shore (fetch) 125
How do I do a calculation with the Math node 228
How do I draw a portal frame 240
How do I draw a section through the ground and excavation 247
How do I draw an elevation 241
How do I draw an excavation 246
How do I draw the building outline on the plan 234
Drawing an outline 124
Drawing and working 144
Durability and cover (all concrete nodes) 196
DXF layers (Ch. 12) 173
Dynamic response cs cd card 123

E

Earlier versions (Ch. 16) 197
How do I edit a value that arrives by wire 203
Editing and selection 145
Element categories (Ch. 11) 157
Elevations and the work plane 144
Embodied carbon (Ch. 17) 199
Engineer's responsibility and independent checking 194
How do I enter a self-weight in a Loads node 228
How do I enter a single set of factored design loads on a column 215
How do I enter base plate design forces by hand 218
How do I enter column loads by case 215
How do I enter column loads on a foundation 209
How do I enter slab loads 229
How do I enter the basic wind speed myself 233
How do I enter the column top moments 215
How do I enter the distance to the shore myself 235
How do I enter the soil bearing capacity (SBC) 209
How do I enter the water table level 209
ETAG 001 / TR029 anchor data (shown when ETAG design is on) 66
Excavation shapes, footprint and formation level 175
How do I export a report as Word, PDF or text 207
How do I export a Sketch-ET scheme as an IFC model 242
How do I export the ground model to a BIM tool 247

F

How do I feed a column from a beam, slab, IFC or Sketch-ET node 215
How do I feed loads into a slab from a Loads or Math node 231
How do I feed two values into a Math node 202
Feedback, help and the tour 22
How do I fill a beam UDL from a dead-load or imposed-load table 221
How do I fill in a row in a Loads node 227
How do I fill in the project details 205
Filters (Ch. 13) 186
How do I find a better concrete beam size 224
How do I find a lighter or lower-carbon steel section 216

How do I find a lighter steel section 225
How do I find a particular element in the model 243
How do I find alternative designs with lower carbon 212
How do I find an economical slab depth and reinforcement 231
How do I find an optimised RC column size and bars 217
How do I find boreholes near my site 248
How do I find mistyped spot heights 246
How do I find the overall horizontal wind force and the worst suction 237
How do I find the wind pressure on a building at a given postcode 232
First launch (Ch. 1) 5
How do I fix a beam that needs compression steel 223
How do I fix a slab deflection failure 231
How do I fix points that have no level 246
Flexure (EC2 6.1, 3.1.7, 5.5) 79
Flexure (EC2 6.1) 109
How do I fold a sticky note 205
Footings (Ch. 10) 146
Formation level (Ch. 12) 175
Foundation (Ch. 14) 191
Foundation (Ch. 16) 195
Foundation / concrete 66
Foundation types and how they are modelled 25
Foundations, columns and base plates (App. A) 208
Foundations, columns and base plates (App. B) 252
Funnelling (Ch. 9) 125
Funnelling and analysis options card 122

G

Gating - a report prints only what the design contains 190
How do I generate a report for one design 206
Geometry (Ch. 3) 26
How do I get a base plate report 220
How do I get a calculation note for a beam, slab or other design 226
How do I get a column report 217
How do I get the bar bending schedule 213
How do I get the embodied carbon of a model 243
How do I get the example workspace back 205
How do I get the programme to size the members to a target utilisation 240
How do I go back to reading the wind speed from the map 233
Ground model (TIN) 174
Groundwater (Ch. 3) 31
How do I group nodes 203

H

How do I hand Sketch-ET column reactions to Column, Foundation or Base plate nodes 242
How do I handle a concentrated load peak in a slab 230
Header status (Ch. 12) 171
Help (Ch. 1) 8
Highlights (Ch. 16) 195
House style (Ch. 14) 189

I

IFC model (Ch. 12) 176
IFC viewer (Ch. 14) 192
IFC viewer and Cut & Fill 196
How do I import a survey into Cut and Fill 245
Import summary (Ch. 12) 173
Importing an IFC model 124
Importing the DXF 172
How do I include or exclude beam self-weight 222
How do I include or exclude categories from the carbon totals 244
How do I include or leave out friction in the shear check 219
How do I include shear deformation in a beam analysis 222
How do I include surface friction or a net pressure coefficient 236
Inputs - pad 26
Inputs (Section 4.4.1) 50
Inputs (Section 5.3) 64
Inputs (Section 6.5.1) 78
Inputs (Section 6.6.1) 82
Inputs (Section 7.1.3) 96
Inputs (Section 7.4.2) 102
Inputs (Section 8.2) 104
Inputs (Section 9.3) 120
Inputs and outputs 101
Inputs column (left) 159
Internal pressure (Ch. 9) 126
Into a beam (the IN dot) 85
Into a slab 112

K

How do I keep a copy of my SI library or move it to another computer 248
How do I keep several beam designs in one Beam node 220
Known limitations (Ch. 11) 165

L

Label matching (Ch. 12) 172
How do I lay out the anchors 219
Layer 1: automatic browser storage 18
Layer 2: project files (.json) 19
Layout of the module 184
How do I leave trees or services out of the ground model 246
Length and buckling inputs 49
How do I let Civil ET apply the load factors to my column loads 215
Libraries and the unit chain 99
Library and privacy 187
How do I link a column to a foundation 214
How do I list the openings in a building 236
Live data (Ch. 14) 189
Live load sets 21
Live sync (Ch. 2) 15
How do I load a PDF, DXF or image of the architect's plan 238
How do I load a second IFC model or replace the first 242
How do I load an IFC model onto the map 201
Load cards (Ch. 3) 27
Load combinations (EN 1990) 77
Load model (load take-down) 47
Loading mode (Ch. 6) 77
Loads (Ch. 8) 105
Loads node (Ch. 7) 99
How do I look at the code charts for my design 237
Lower, default, upper bound 161

M

How do I make a copy of my project 205
How do I make web bearing count in a steel beam check 225
Making, reshaping and removing wires 14
Manual additions (Ch. 11) 163
Map layers (Ch. 13) 185
Material buckets (Ch. 11) 158
Materials card (Ch. 3) 26
Materials, cover and fire 106
Math node (Ch. 7) 102
How do I measure and dimension on the plan 241

How do I measure distances and areas in the IFC viewer 243
How do I measure the distance to the shore automatically 234
Member, supports and loads 75
Method (Ch. 11) 160
How do I minimise or expand a card 203
How do I model a duopitch or hipped roof 233
How do I model a monopitch roof 233
Moment redistribution and K' 109
Mouse and trackpad 16
How do I move a card and line it up with others 203
How do I move or delete a support or load on a beam 221
How do I move, copy or rotate members precisely 241
Moving around the map 16
Multiple foundations and the Foundation node 33
How do I mute a load component on a slab 229
How do I mute or detach wired loads on a beam 226

N

Nearby boreholes (Ch. 13) 186
Node map and workspace 195
Notes (Ch. 16) 197
Notes on use 200

O

One source per design 14
How do I open a base plate design and add more 218
How do I open a module from its card 208
How do I open the Blue Book website from a Steel sections card 227
How do I open the Column node and switch between steel and concrete 214
How do I open the Foundation designer 208
Opening and adding a base plate 64
Opening and adding a beam 74
Opening and adding a column 47
Opening and adding a slab 104
Opening it (Ch. 10) 141
Opening the designer 24
Opening the node 119
Openings and internal pressure card 123
Optimiser (Ch. 8) 111
Ordnance Survey and other data sources 199
Other inputs (Ch. 3) 28
Other structures card 123
Out of a beam (the OUT dots) 85
Out of a slab 112
Output and wiring 102
Outputs (Section 12.10) 177
Outputs (Section 3.6) 32
Outputs (Section 9.7) 127
Outputs and ports 98
Outputs and results panels 111
Outputs, ports and wiring 164
Overall force and the worst local suction 126
How do I override the stiffness of a beam 222
Overturning (EQU) and eccentricity 30
Overview (Section 11.1) 156
Overview (Section 12.1) 171
Overview (Section 14.1) 188
Overview (Section 2.1) 12
Overview (Section 3.1) 24

P

Pad (PF) 25
How do I pause or resume one wire 202
Performance and level of detail 22
Persistence (Ch. 11) 157
Pile cap (PC) 25
How do I pin a sticky note to a card or group 204
How do I place columns, beams and slabs 239
How do I place the model on the OS grid and show a Google map behind it 242
How do I place the site by clicking on the wind map 232
Plain concrete or reinforced 31
Plan - zones 126
Plan outline card 121
Plate finite element (default) 108
Point and line loads 106
Point review (Ch. 12) 173
Portal frames (Ch. 10) 145
Ports, wires and links 13
How do I produce a combined report of several designs 207
How do I produce a foundation report 213
How do I produce the Cut and Fill report 248
How do I produce the IFC report 245
How do I produce the wind report 238
Producing a report 188
Project and global settings 21
Project details and RIBA stages 21
Punching Shear and Calculation Options card 28
Punching under point loads (EC2 6.4) 110
How do I push data through a wire manually 202
How do I put a report on the map when no location was found 248

Q

Quantities and carbon 52
Quantities and embodied carbon (Section 6.8) 84
Quantities and embodied carbon (Section 8.8) 112
Quantity output ports 164

R

Re-classifying categories and elements 162
How do I read a Loads answer in newtons instead of kilonewtons 228
How do I read design actions rather than service actions on a beam 222
How do I read the base plate results 220
How do I read the cut, fill and import or export volume 247
How do I read the full wind calculation step by step 237
How do I read the quantities and embodied carbon of a beam 225
How do I read the steel column checks 216
How do I read the summary of checks and utilisation 210
How do I read the worst utilisation on the Foundation card 214
Reclassification and reinforcement allowance 161
How do I recover when Civil ET fails to start 208
Reference documents (Ch. 7) 101
How do I refresh a report after changing my design 207
Reinforced concrete columns (EN 1992-1-1) 50
Reinforced design (EC2) 31
Reinforcement (Ch. 8) 107
Reinforcement allowance (Ch. 11) 161
Reinforcement card (shown while reinforcement design is on) 28
How do I remove a wire 201
How do I remove all wires at once 201
How do I remove long thin triangles over unsurveyed ground 246
How do I rename a node 203
How do I rename, switch or delete a load set 227
How do I reorder or collapse the input cards 210
How do I replace or remove an underlay 239

How do I replace the reinforcement quantity with my own bar-bending-schedule number 225
Report (Section 11.14) 166
Report (Section 12.11) 177
Report (Section 5.6) 68
Reports (Ch. 16) 196
Reports (Ch. 2) 19
Reports (Section 3.8) 33
Reports (Section 8.7) 111
Reports (Section 9.8) 128
Reports and exports 148
Results (Section 10.9) 147
Results (Section 5.5) 68
Results (Section 6.6.3) 83
Results (Section 9.6.4) 127
Results and reports 52
Results, reports and the node card 85
How do I return to the node map from a module 208
How do I review contested survey levels 246
Review issues ("code scope") 32
Reviewing a report 186
RIBA stages (Ch. 10) 148
Ribbon tabs (Ch. 10) 142
Right panel (Ch. 11) 160
Roofs (Ch. 9) 125
How do I rotate the building to match the plot 234
Row fields (Ch. 7) 99
How do I run a full anchor check to ETAG 001 / TR029 219

S

How do I save my project and open it again 205
How do I save, advance to or load a RIBA stage snapshot 206
Saving, loading and exporting 18
SCORS rating (Ch. 11) 164
Screen layout (Ch. 3) 24
Screen reference - panel by panel (Ch. 10) 149
Screen reference - panel by panel (Ch. 11) 166
Screen reference - panel by panel (Ch. 12) 178
Screen reference - panel by panel (Ch. 4) 54
Screen reference - panel by panel (Ch. 5) 70
Screen reference - panel by panel (Ch. 6) 86
Screen reference - panel by panel (Ch. 8) 113
Screen reference - panel by panel (Ch. 9) 129
Screen reference - panel by panel (Section 3.10) 34
Section and material 49
Section families (Ch. 7) 97
Section properties and resistance routes 97
Sections (Ch. 12) 176
How do I see column reactions and member diagrams 241
How do I see section properties, fire parameters or resistance tables 226
How do I see the black and white detail drawing 213
How do I see the full working for a check 210
How do I see the lower, default and upper carbon totals 244
How do I see the model in 3D 241
How do I see the quantities for each storey 243
How do I see the version and contact details 206
How do I see the wind flow around the building in 3D 237
How do I see the wind pressure on every wall and roof zone 237
How do I select several nodes 203
Selecting, moving, aligning and grouping 15
Self weight and older load nodes 101
Self-weight (Section 4.2.4) 49
Self-weight (Section 6.4.5) 78
How do I send a beam, slab, column or foundation from the model to a design node 245
How do I send a column's load to a particular foundation design 202
How do I send a Loads node, Math result or IFC element to a beam 226
How do I send a Sketch-ET scheme to the IFC node for a carbon study 242
How do I send base plate size and utilisation to the foundation 220
How do I send beam reactions to a column 226
How do I send column loads to a base plate 218
How do I send column loads to a foundation 218
How do I send feedback 206
How do I send IFC quantities to a Math node 245
How do I send slab loads to a beam 226
How do I send the wind force to the Beam, Slab, Column, Foundation or Base plate node 238
Serviceability (EC2 7.4) 81
How do I set a one-way slab as an end span with continuity 230
How do I set bars and links on a concrete beam 223
How do I set cover for an RC column 217
How do I set cover, exposure class and cover mode on a concrete beam 223
How do I set embodied carbon options 213
How do I set fire resistance for an RC column 217
How do I set lateral restraint on a steel beam 224
How do I set materials, cover and fire rating on a slab 230
How do I set quantities and carbon for a slab 231
How do I set steel column effective lengths and buckling 216
How do I set the altitude of the site 232
How do I set the base plate size and grade 219
How do I set the building roof type 233
How do I set the C1 factor for lateral-torsional buckling 225
How do I set the column and weld on a base plate 219
How do I set the column quantity and carbon factors 218
How do I set the concrete grade and design working life 211
How do I set the crack-width limit and aggregate size 211
How do I set the dynamic factor cs cd 236
How do I set the effective span of a slab 229
How do I set the exposure class and cover 211
How do I set the formation level 247
How do I set the foundation under a base plate 220
How do I set the gross internal area and get a SCORS band 245
How do I set the lateral stability choice and the floor area 240
How do I set the lateral-torsional buckling moment shape 216
How do I set the number of storeys and storey height 239
How do I set the RC column effective length and braced or unbraced behaviour 217
How do I set the reinforcement bars or mesh 211
How do I set the reinforcement of a slab 230
How do I set the scale of the underlay 239
How do I set the site from an OS grid reference 232
How do I set the site from latitude and longitude 232
How do I set the soil parameters for sliding and settlement 209
How do I set the terrain to town or country 234
How do I set the transport and construction stages (A4 and A5) 244
How do I set up the column grid 239
Setting up (Ch. 10) 142
Setting-out grid (Ch. 10) 143
Settings in the import panel 173
Settlement (EC7 Annex F.2) 31
Several models (Ch. 11) 156
Shapes (Ch. 12) 175
Shear (EC2 6.2.2) 109
Shear (EC2 6.2) 80

How do I show only clay at 1.5 m depth 248
How do I show only one storey or one category in the 3D viewer 243
How do I show or hide the floating cards in a viewer 237
SI map (App. A) 248
SI map (App. B) 259
Site and wind climate card 120
Site location and the Google 3D map 147
Site position and OS grid 125
How do I sketch the building over a PDF or image of the plan 234
Sketch-ET (Ch. 16) 196
Sketch-ET (preliminary scheme) 192
Sketch-ET link (Ch. 11) 165
Slab (Ch. 14) 192
Slab (Ch. 16) 196
Slab system, loads and defaults 143
Slab type, geometry and supports 104
Sliding (EC7 6.5.3) 30
How do I snap support positions to a grid or draw an angled beam 221
Snapping (Ch. 10) 144
Sockets (Ch. 2) 13
How do I split the windward wall into strips for a tall building 236
How do I start a new project 205
How do I start a Sketch-ET design 238
How do I start from a template building or portal frame 240
Start-up crash guard and backups 23
Steel beam design (EC3) 82
Steel columns (EN 1993-1-1) 49
Steel design guidance 199
Steel sections (Blue Book) 96
Sticky notes (Ch. 2) 20
Stiffness: E and I 76
How do I stop a foundation taking data from the column 214
How do I stop the site moving when I click the wind map 232
Strip footing (SF) 25
Surface display (Ch. 12) 174
How do I switch a foundation header between one foundation's carbon and the total 207
How do I switch a load group in or out of the checks 210
How do I switch between curved and straight wires 202
How do I switch between plan, section and 3D views 213
How do I switch internal pressure off 236
System requirements (Ch. 1) 5

T

How do I take base plate forces from a Column node 219
How do I take the guided tour 206
Templates (Ch. 10) 146
Tension-side axial: NEd,T 68
Terrain and exposure card 122
The 15 node types at a glance 8
The calculation and units 102
The combined (node map) report 192
The example workspace 19
The header (Ch. 1) 6
The heat layer and its check 187
The input, results and drawer panels 7
The map (Ch. 13) 185
The module screen 159
The modules (Ch. 1) 7
The no-redesign-unless-numbers-change rule 33
The Report button 188
The report window 189
The ribbon and tool strips 8
The right panel on the node map (Ch. 1) 8
The right panel on the node map (Section 2.7) 17
The screen (Ch. 10) 141
The SI map card 183
The structural model 75
The summary panel 187
Timber beams (Ch. 6) 83
Tools (Ch. 10) 144
Total (Ch. 11) 163
Touch and pen (iPad, Android and Windows tablets) 17
How do I treat every wind direction as having a directional factor of 1.0 235
How do I turn a pressure into a force with Math 228
How do I turn an area load into a load per metre or a force in a Loads row 227
How do I turn Auto sync on or off 202
How do I turn on reinforced design for a pad 210
Two ways to enter loads 47
How do I type a building footprint instead of drawing one 247
How do I type my own loads on a wired column 215

U

UK NA 5.1.3(1)P option (c) and load arrangements 109
ULS expression (Ch. 6) 77
Underlay (Ch. 10) 142
How do I undo a change in the foundation designer 214
How do I undo an edit or start again with a corrected DXF 247
Undo and redo 18
How do I undo and redo 204
How do I undo and redo in Sketch-ET 241
How do I undo or redo a change in a beam 221
Unit checks (Ch. 2) 14
Units (Ch. 12) 173
Uplift (EC7 2.4.7.4, UK NA) 30
How do I upload an IFC model and see its quantities 242
How do I use a Dimensions node 228
How do I use a manufacturer's EPD in the carbon calculation 244
How do I use a net bearing allowance 209
How do I use a Steel sections node to set a column section 227
How do I use frame mode for steel buckling lengths 216
How do I use moment redistribution or the ductility cap on a beam 223
How do I use pre-factored loads on a beam 222
How do I use the 20 % support redistribution option in a slab 230
How do I use the carbon panel and SCORS rating 207
How do I use the node map with a finger or pen 204
How do I use the wind force as a value in a Math node 238

V

Version 1.2.0 (Ch. 16) 195
How do I view the RC column interaction diagram and 3D view 217
How do I view the site on a Google aerial or 3D map 238
How do I view the slab in 3D and see a reinforcement heat map 231
Viewer tabs (middle) 159
Viewer toolbar (Ch. 11) 159
Views (Ch. 9) 126
Views and the detail drawing 32
Volume method (Ch. 12) 176

W

How do I walk through the model 243
Walls (Ch. 9) 125
Warnings (Ch. 9) 126

What a stage switch does 21
What a wire carries 100
What changed in v1.2.0 70
What Civil ET is 4
What counts as a level (text labels) 172
What each module's report contains 191
What feeds the rows (wires) 48
What is new in version 1.2.0 4
What is read from the file 156
What it does (Section 10.1) 141
What it does (Section 7.1.1) 96
What it does (Section 7.2.1) 99
What it does (Section 7.3.1) 101
What it does (Section 7.4.1) 102
What it does (Section 9.1) 119
What the calculation does (Section 5.4) 66
What the calculation does (Section 9.5) 124
What the diagrams show, and what is designed 78
What the parser reads 172
What the receivers do 100
What the steel design does not cover 83
Which factor applies (precedence) 160
Which pairs can be connected 13
Which wires are allowed (Ch. 1) 11
Which wires are allowed (Section 7.5) 103
Whole-site total (Ch. 12) 176
Wind (Ch. 14) 192
Wind designer (Ch. 16) 196
Wind directions and factors card 121
Wind map (Ch. 9) 126
Wind, Sketch-ET, IFC and Cut & Fill (App. A) 231
Wind, Sketch-ET, IFC and Cut & Fill (App. B) 256
How do I wire a Loads node to a beam, slab, foundation or Math node 228
How do I wire a slab to a beam and on to a column 231
Word, text and PDF 189
How do I work with load sets 208
Workspace carbon summary and SCORS rating 164
Workspace, node map, projects and reports (App. A) 201
Workspace, node map, projects and reports (App. B) 250

Z

How do I zoom, pan and fit the map 204