OpenPT

User Manual

Post-tensioned and reinforced concrete slab analysis and design
OpenPT .optp projects · ADAPT, SAFE and ETABS exchange · TCVN 5574:2018, EC2, ACI 318, SP 63

Edition for the build of 2026-11-25 (feature specifications 01–306)
Document date: 2026-11-25

Contents

1 Getting started

- 1.1 Installation
- 1.2 The main window
- 1.3 The 3D view
- 1.4 Tab Focus
- 1.5 Start view and what is drawn
- 1.6 Properties palette
- 1.7 Interface language
- 1.8 Help ▾
- 1.9 Display units
- 1.10 Long runs, stability and recovery
- 1.11 Dialogs
- 1.12 Results display basics
- 1.13 Keyboard Shortcuts

2 Files and projects

- 2.1 File types
- 2.2 Creating a new model
- 2.3 Importing a model
- 2.4 Saving: .optp only
- 2.5 Sidecar files
- 2.6 Results store
- 2.7 Locking the model
- 2.8 Single-file projects (.optp)
- 2.9 Exporting
- 2.10 Exporting to Revit, IFC and AutoCAD
- 2.11 File association

3 Building and editing a model

- 3.1 Adding structural components
- 3.2 Supports
- 3.3 Layers
- 3.4 Snapping
- 3.5 Selecting items
- 3.6 Grips and the item context menu
- 3.7 Item Properties
- 3.8 Modify Selection (batch edit)
- 3.9 Delete, Undo, Redo, Table edits
- 3.10 Draw tools (construction geometry)
- 3.11 Grid lines
- 3.12 Modify tools
- 3.13 Columns and walls above the slab from the ones below
- 3.14 Vertical copy / move (to other planes)
- 3.15 Trim/Extend Tendon
- 3.16 Parallel and radial support lines
- 3.17 Beam support lines
- 3.18 Column positions near drop panel edges (meshing note)

4 Multi-level models

- 4.1 Single-Level and Multi-Level mode
- 4.2 Story Manager
- 4.3 Copy Storey

4.4 Analysis of a multi-level model
4.5 Saving and exporting multi-level models

5 Command-line use

5.1 Files and projects
5.2 Analysis and design
5.3 Pictures
5.4 Developer and diagnostic switches
5.5 Self-tests

6 Tendons

6.1 Where the tendon tools are
6.2 Adding a tendon
6.3 The tendon window (Tendon Editor)
6.4 Profile geometry
6.5 Friction, anchor seating and elongation
6.6 Long-term losses
6.7 Equivalent loads of a tendon
6.8 Prestress loading on the analysis model
6.9 Hyperstatic (secondary) actions
6.10 Tendon display (Tendon Display Manager)
6.11 Tendon layout helpers
6.12 Chair heights
6.13 Trim/Extend Tendon
6.14 Elongation Table

7 Meshing and analysis

7.1 Workflow
7.2 Mesh Generation dialog
7.3 How the mesh is built
7.4 Node shift
7.5 Members in the analysis model
7.6 Loads on the mesh
7.7 The finite-element solver
7.8 Speed
7.9 Modal analysis
7.10 Time-dependent analysis

8 Loads, criteria and combinations

8.1 Load tools (Loads ▸ Add Loads)
8.2 Load cases (Loads ▸ Define ▸ Load Cases)
8.3 Design criteria and design codes
8.4 Load combinations (Loads ▸ Define ▸ Load Combinations)
8.5 TCVN total deflection

9 Results

9.1 Showing a result
9.2 Results View pane
9.3 Contours
9.4 Strip moment diagrams
9.5 Diagrams
9.6 Display tab
9.7 Section actions
9.8 Section Cut (View ▸ Section)
9.9 Design section information panel

10 Design

10.1 Design strips: support lines, tributaries and sections
10.2 Strip design (Design the Sections)
10.3 Base rebar (Tendons & Rebar tab)
10.4 Generate Rebar and Rebar Options
10.5 Cracked deflection
10.6 Punching shear
10.7 Beams without a slab (the Beam tab)
10.8 Beam design (Generate Beam Rebar)
10.9 Section Check
10.10 Time-Dependent Analysis (ACI 209)
10.11 Calculation Report (.docx)
10.12 Reports and the Report Compiler
10.13 Sending a floor to ETABS

11 Verification against ADAPT

12 Working with SAFE models

12.1 1. Importing a SAFE file
12.2 2. What OpenPT reads from a SAFE text file
12.3 3. Behaviour of the converted model
12.4 4. Sidecar files: openings and columns
12.5 5. What goes where
12.6 6. Working with the imported model
12.7 7. Going back to SAFE, ADAPT and ETABS
12.8 8. Older SAFE-kind documents (legacy)
12.9 9. Limitations of the SAFE path

13 TCVN 5574:2018 design

13.1 1. Total deflection F = f1 − f2 + f3
13.2 2. TCVN flexural design of strips and sections
13.3 3. TCVN punching shear without reinforcement (8.1.6)
13.4 4. TCVN punching shear with calculated reinforcement (8.1.6.2.2, 8.1.6.3.2)

14 Worked example: a SAFE floor from import to TCVN design

14.1 Step 0. Prepare the folder
14.2 Step 1. Import the file
14.3 Step 2. Check the model
14.4 Step 3. Mesh and analyse
14.5 Step 4. Check the analysis
14.6 Step 5. TCVN total deflection
14.7 Step 6. Slab reinforcement
14.8 Step 7. Punching shear with reinforcement
14.9 Step 8. Report and save
14.10 Step 9 (optional). Continue in ADAPT, SAFE or ETABS

15 Known limitations

16 Glossary

17 Standards referenced

About this manual

This User Manual is the complete reference of OpenPT. To learn the program step by step, start with the separate **OpenPT User Guide** (tutorials). This manual has two kinds of content:

- **Procedures**: the steps for each command, in the chapter for that function.
- **Reference**: the option tables, defaults, formulas, code clauses, output-file columns and command-line switches. These sit under headings such as *Options*, *Method* and *Output*.

Feature numbers such as "spec 91" refer to OpenPT's internal feature specifications. They are given so that a reader can find the full definition of a rule when this manual is too brief.

Conventions

Convention	Meaning
Bold	A button, menu item, tab or dialog title as it appears on screen
`Monospace`	A file name, folder, command-line switch, or a value you type
Tab ▸ Group ▸ Button	A path through the ribbon, for example **Analyse ▸ Run ▸ Run Analysis**
> note	A practical warning or tip

Units follow the model: an imported file keeps its own unit set, and the display units can be changed in the status bar. Design to TCVN works internally in N and mm, and its results are reported in kN, kN·m, mm and MPa.

What OpenPT does

OpenPT analyses and designs concrete floor slabs, flat or with beams, prestressed with unbonded or bonded tendons, or reinforced only. It meshes the floor with shell elements, solves it by the finite element method, and designs reinforcement along design strips and at columns.

Model sources. The native file is the single-file project `.optp` (**File ▸ Open**). **File ▸ Import ▸ Import model...** reads an ADAPT-Builder `.INP` or `.adm`, a SAFE `.F2K`, a SAFE `$sf` or a SAFE `.FDB` into a new untitled model, which you edit, mesh, analyse and design to TCVN 5574:2018, SP 63, EC2 or ACI 318 (14 and 19) with the same tools. The model is saved as `.optp` and exported to `.INP`, `.adm`, `.F2K`, `.FDB`, IFC, DXF, Revit or ETABS.

Main capabilities:

- **Tendons:**
- friction and seating losses;
- long-term losses;
- profiles;
- equivalent (balanced) loads;
- hyperstatic actions;
- the theoretical elongation table for jacking control (Excel and PDF).
- **Meshing and analysis:** automatic quadrilateral meshing; single-level and multi-level (several storeys in one model) analysis; load cases and combinations.
- **Results:**
- deflections;
- plate moments and forces;
- section cuts and design-section actions;
- long-term cracked deflection;
- TCVN total deflection f = f1 − f2 + f3.
- **Design:**
- strip design for PT and RC (stress, strength and minimum steel);
- base reinforcement;
- the calculated rebar plan;
- punching shear: EC2 and ACI with studs or stirrups, and TCVN without and with calculated reinforcement.
- **Project handling:**
- a results store, so that a solved model reopens without re-analysis;
- a model lock that protects results from accidental edits;
- a single-file project (`.optp`);
- grid lines with bubbles, spacing dimensions and snapping;
- export of the model to Revit (IFC) and of a level plan to AutoCAD (DXF);
- an English or Vietnamese interface;
- beams without a slab, with beam design and a Section Check calculator (SP 63, TCVN 5574:2018, EC2 and ACI 318);
- modal and time-dependent analysis;
- a Word calculation report and the Report Compiler.

Typical workflow

```
Open (.optp) or Import model (.INP / .adm / .F2K / .FDB)
   │
   ├─ check / edit geometry, supports, tendons, loads, strips
   ├─ mesh  ──►  analyse  (results store written, model locked)
   ├─ review results: deflection, actions, section cuts
   ├─ design: strips, base rebar, calculated rebar plan, punching
   ├─ TCVN checks: total deflection, punching with reinforcement
   └─ save: .optp; export .INP / .adm / .F2K / .FDB, CSV, Excel, PDF, IFC, DXF, report
```

1 Getting started

OpenPT is a Windows desktop program for the analysis and design of post-tensioned (PT) and reinforced-concrete (RC) floor slabs. Its own project file is the `.optp`. It imports ADAPT-Builder exchange files (.INP) and project files (.adm), SAFE text models (.F2K, .$sf) and SAFE databases (.FDB), and it exports models to those formats, to Revit, to IFC and to AutoCAD. This chapter explains how to start the program and describes the main window.

1.1 Installation

OpenPT is installed with its Setup program (`Setup.exe`). The program is self-contained: no separate .NET runtime has to be installed. If Autodesk Revit 2024 or later is on the computer, Setup also copies the OpenPT add-in for Revit (see "Exporting to Revit, IFC and AutoCAD"); restart Revit once afterwards.

Item	Value
Program file	`OpenPT.Gui.exe`, in the folder chosen in Setup
Documentation	The folder `docs\` beside the program file holds the licence, this manual, the User Guide and the two verification reports (see "Help ▾" below)
User settings	`%APPDATA%\OpenPT\` (for example `plan.ini` for view and snap settings, `recent.txt` for the recent-files list, `shortcuts.txt` for your keyboard shortcuts)
Error logs	`error.log` in the same folder (it rolls over at 2 MB to `error.old.log`); `hang.log` and `timing.log` stand beside it (see "Stability and recovery")
Working folders of projects	`%LOCALAPPDATA%\OpenPT\projects\` (see "Single-file projects (.optp)")

To start OpenPT:

1. Double-click `OpenPT.Gui.exe`, or use the shortcut that Setup made.
2. To open a file at start-up, give its path as the only argument, for example `OpenPT.Gui.exe "D:\Jobs\B01\B01.optp"`. A path to an `.optp` file opens it. A path to any other model file (.INP, .adm, .F2K, .$sf, .FDB) imports it as a new untitled model, the same as dropping the file on the window (see "Files and projects"). A first argument that starts with `--` runs a command-line function instead (chapter "Command-line use").

The window opens on the Model tab with no document. Without a licence, a ship build shows the licence dialog and does not run (see "Help ▾").

1.2 The main window

From top to bottom, the main window has these parts:

```
[icon] [New] [Open] [Save] [Undo] [Redo]   <document name>*                          [–]
[□] [×]
[File] Model  View  Loads  Tendons & Rebar  Analyse  Design  Report  Beam    Level [combo]
▲ ▼ [Single][Multi]  Help ▾
ribbon groups of the selected tab
properties/results panes (right) | Plan  3D   ...                    Load: [combo]
command bar:  prompt text ....................................  Point: [ x,y ]
status bar:  file | counts | Check | state | run bar | ... | cursor | scale | Units ▾
```

There is **one ribbon** for every document: the same tabs and groups whether the model came from an ADAPT file, from a SAFE file or was drawn in OpenPT. Where a command does not apply to the open model, it is greyed.

Title bar and quick access toolbar

The window draws its own title bar. From the left it holds the application icon, the **quick access toolbar**, the title text, and the minimise, maximise/restore and close buttons. The quick access toolbar has **New model** (Ctrl+N), **Open** (Ctrl+O), **Save** (Ctrl+S), **Undo** (Ctrl+Z) and **Redo** (Ctrl+Y). The title text is the document name, followed by `*` when the model has unsaved changes, and by a lock mark when the model is locked (see "Locking the model"). Dragging the title bar, double-clicking it, snapping the window to a screen edge and the system menu work as in any Windows program. Dialogs keep the standard Windows title bar.

The File menu

The **File** button at the start of the tab row opens a menu:

New model... (Ctrl+N), *Open...* (Ctrl+O), *Recent files* ▸ (last 8 files), *Save* (Ctrl+S), *Save As...*, *Import* ▸ (*Import model...*, *Import DXF/DWG...*), *Export* ▸ (the list in "Exporting"), *Exit*.

Ribbon tabs and groups

After the File button come eight tabs, in work order. The window opens on **Model**; the last tab you chose is remembered between sessions.

Tab	Groups
Model	Levels & Grids, Selection, Edit, Snap, Draw, Add Structural Components, Modify, Convert
View	Zoom, Views, Camera, Display, Layers, Section, Table
Loads	Define, Add Loads, Show Loads, Load Pattern
Tendons & Rebar	Settings, Model, Modify, Shop Drawing, Visibility, Rebar Criteria, Base Rebar, Generate, Rebar Visibility
Analyse	Meshing, Run, Punching, Modal, Check, Results
Design	Criteria, Design Options, Strip Modelling, Strip Layers, Section Design
Report	Print, Compiled Reports, Default Report, Analysis Reports, Export, ETABS
Beam	Model, Loads, Tendon, Analysis, Results, Design

Groups that do not apply to the open document are hidden or greyed: the **Convert** group of Model (it holds **Convert to OpenPT Model**, for a document of the older SAFE kind only), the four F2K view choices of **View** ▸ **Views** (**Model**, **Results**, **Tables**, **Strip plan**), the **Design Options** group, the small

Save button of **Analyse** ▸ **Run** (it saves the results of an F2K-kind document; with an INP-kind model it is hidden), the **Table** group of View (only with the advanced setting "Show INP Table/Text tabs"), and the tendon groups of the Tendons & Rebar tab (on a model whose source has no tendon commands the tab shows one sentence that says so). The **Results** group of Analyse is one group for both kinds of document: with an INP-kind model it holds the four buttons of the Results View (see "Showing a result"); with an F2K-kind model it holds the result choosers. Every ribbon control has a text label written in whole words (no clipped word and no abbreviating dot; the full command name is in the tooltip), and the groups fit a window 1600 pixels wide without a scroll arrow.

Drop-down buttons. The snap kinds of the Model tab and the layers of the View tab are not separate buttons. **Snap** ▾ (Model ▸ Snap) and **Layers** ▾ (View ▸ Layers) are labelled drop-down buttons, and so is **Look** ▾ (View ▸ Display). A path such as **View** ▸ **Display** ▸ **Look** ▾ ▸ **Group Colours** means: the View tab, the Display group, the **Look** ▾ list, the item *Group Colours*; **Model** ▸ **Snap** ▾ ▸ **Ortho** is the item *Ortho* of the Snap list. What each list holds is described in "Layers of the View tab" and "Snapping".

A path in this manual such as **Analyse** ▸ **Run** ▸ **Run Analysis** means: the Analyse tab, the Run group, the Run Analysis button. A path that starts with **File** ▸ is in the File menu, and one that starts with **Help** ▾ ▸ is in the Help menu.

Where the main commands are.

Command	Place
Story Manager, Quick Grids..., Grid Systems...	Model ▸ Levels & Grids
Select by Type, Window Selection/Pick Mode, Pan, Rotate, Layer Settings	Model ▸ Selection
Delete, Properties Palette, Move to Layer	Model ▸ Edit
Snap On/Off (**Snap** ▾), the snap kinds and Ortho (the items of its list), Snap Settings...	Model ▸ Snap
Draw tools	Model ▸ Draw
Slab, Wall, Column, Drop Panel, Beam, Support, Opening	Model ▸ Add Structural Components
Modify Selection, Item Properties, Copy ▾, Mirror, Scale, Copy Up (Columns & Walls Above from Below), Delete Point, Add Point (Remove Point, Insert Point)	Model ▸ Modify
Zoom Extents (F), Zoom In, Zoom Out, Redraw	View ▸ Zoom
Plan / 3D / Plan + 3D	View ▸ Views ▸ View
Settings (View Settings), Show by Type..., Colour Palette... (large buttons); **Look** ▾ (Grid, Tab Focus, Show Details, Group Colours); Shaded, Default Display (small buttons)	View ▸ Display
Layers ▾ (Labels, Drawing, Grid Lines, Grid Dimensions, the parts, Tendons, Tendon Heights, Loads, Rebar, Mesh, All Layers, Model Only); the toggles Components, Tendons, Rebar, Loads and Mesh	View ▸ Layers
Load Patterns, Load Cases, Load Combinations, TCVN deflection...	Loads ▸ Define
Tendon Editor, Trim/Extend Tendon	Tendons & Rebar ▸ Modify
Elongation Table...	Tendons & Rebar ▸ Shop Drawing
Generate Rebar, Rebar Options...	Tendons & Rebar ▸ Generate

Command	Place
Mesh Generation, Cell size (mm)	Analyse ▸ Meshing
Run Analysis (F9), Analysis Options..., Run Summary, Cracked Deflection, Time-Dependent Analysis..., Lock/Unlock Model	Analyse ▸ Run
Shear Design Options, Shear Check, Punching bars...	Analyse ▸ Punching
Modal Analysis Options..., Run Modal Analysis	Analyse ▸ Modal
Check Model (F5)	Analyse ▸ Check
Save results (F2K-kind documents)	Analyse ▸ Run (**Save**)
Results Display Settings (the Results View pane), Results Scale Down, Default Results Scale, Results Scale Up	Analyse ▸ Results (INP-kind documents)
Results group, Result step, Slab component, Beams component, Clip shell range, Deformed shape (3D)	Analyse ▸ Results (F2K-kind documents)
Design Code..., Materials, Allowable Stresses	Design ▸ Criteria
Support line, splitter, tributary tools, Auto Strips...	Design ▸ Strip Modelling
Generate Sections, Use Existing, Design the Sections, Section Check...	Design ▸ Section Design
Full Report, Report Compiler	Report ▸ Compiled Reports
Send to ETABS..., Get ETABS Forces..., Import ETABS Forces from File...	Report ▸ ETABS
Generate Beam Rebar, Beam Rebar Options..., Show Beam Rebar	Beam ▸ Design

Right end of the tab row

Control	Use
Level combo	Selects the level (reference plane) shown in the Plan and 3D tabs. The closed box shows the level's name and elevation, for example "Level 2 (z = 3.30 m)", or the level's own name; the words Top plane, Current plane and Bottom plane are never shown as names. On a document of the older SAFE kind the box reads "Level 1" (no elevation) and is disabled, and the **Level Up**, **Level Down**, **Single-Level** and **Multi-Level** buttons are hidden.
Level Up / **Level Down**	Shows the plane above / below the shown one (by elevation). Disabled at the top / bottom plane.
Single-Level / **Multi-Level**	Two toggle buttons; see the chapter "Multi-level models".
Help ▾	*Documentation...*, *Associate .optp files with OpenPT*, *About OpenPT*, *Licence...*, *Keyboard Shortcuts...*, and the two language items *English* and *Tiếng Việt* (see "Help ▾" and "Interface language").

Collapsing the ribbon

Double-click a tab header, or press **Ctrl+F1**, to collapse the ribbon so that only the tab row stays. Clicking a tab while the ribbon is collapsed shows its panel as a drop-down over the drawing; it closes

when you click elsewhere. Double-click a tab again, or press Ctrl+F1, to expand it. The selected tab and the collapsed state are remembered between sessions.

Document tabs

Below the ribbon the drawing area has tabs:

Tab	What it shows
Plan	The plan of the shown level: slabs, openings, drop panels, beams, walls, columns, tendons, loads, strips, rebar, and results (contours and diagrams). Most editing is done here.
3D	A 3D view of the level (or of all levels in Multi-Level mode).
Table, **Text**	The INP table selected in the model tree, and the INP file as text. Hidden by default; see "Model tree and the Table and Text tabs".
Member forces (G)	For a Multi-Level model, after an analysis.
Modal	After a modal run (see "Modal analysis").

There are no separate Analysis, Tendons or Check tabs and no SAFE tab: results are shown in the Plan and 3D tabs, the tendon list and profiles are in the Tendon Editor and the Properties palette, and the check messages are in the Check Model window (**Analyse** ▸ **Check** ▸ **Check Model**, F5). The **View** button (View ▸ Views) switches the drawing area between **Plan**, **3D** and **Plan + 3D** (side by side; the active pane has an accent line, and a result is drawn only in the active pane: see the chapter "Results"). 3D and Plan + 3D are greyed when no 3D view is possible.

At the end of the tab row, the **Load:** drop-down picks the load combination or load case shown in the Plan and 3D tabs (see "Results View pane"). It is hidden until there is a load to pick.

Panes and bars

- **Properties pane** (right column, **Model** ▸ **Edit** ▸ **Properties Palette**, Ctrl+Shift+P; shown by default, command alias PR). It follows the selection and shows its properties; with no selection it says so ("No document." when none is open). It is stacked above the Results View pane when both are shown. See "Properties palette".
- **Results View pane** (right column): opens after a finished run (you may close it). See the chapter "Results".
- **Model tree** (left pane): see below.
- **Info bar and message bar**, above the drawing: what the last command did, and offers such as "No design sections yet. Generate them now?".
- **Command bar**: on the left, the prompt of the running command and messages (errors in red). On the right, the **Point:** box. While a command waits for a point you can type it here and press Enter:

Input	Meaning
`x,y`	Absolute coordinates (in the plan length unit; in SI, `6,0` is 6 m)
`@dx,dy`	Relative to the last point
`@L<A`	Polar: length L at angle A (degrees from +X), from the last point

Use a point `.` as decimal separator. A wrong entry shows "Type x,y or @dx,dy or @length<angle" and stays in the box for correction. Some commands also ask for a number (radius, number of sides, angle)

in this box. **Esc** ends the command. While the box is empty it shows the grey hint "Enter x,y or a command alias...". Aliases for commands are assigned by you in **Help ▾ ▸ Keyboard Shortcuts...**; OpenPT has no built-in aliases except those named in this manual (for example `LA` for Layer Settings and `PR` for the Properties palette).

- **Status bar**, at the bottom, in plain text segments: the file name (the full path in its tooltip), counts ("3 levels, 44 tendons"), the **Check** state (errors and warnings; a small coloured square), the **analysis state** (**Not analysed** grey, **Analysed - not designed** amber, **Designed** green, **Design out of date** amber, **Analysed (multi-level)** green; clicking it does nothing), the **run bar** while a run is active (see "Long runs"), the number of labels hidden ("n labels hidden, zoom in"), the cursor coordinates, the drawing scale (for example "1 : 150") and the **Units** drop-down (see "Display units"). While the 3D tab is active the cursor segment shows the camera angles, for example `A -45 E 60`.

Model tree and the Table and Text tabs

The left pane is the model tree: the blocks of the INP model (levels, geometry tables, tendons, loads). Selecting a node shows that table in the **Table** tab, if that tab is shown. The tree names its blocks in plain words, for example "File information", "Levels", "Mild steel materials", "Drop caps and panels" and "Combination 3"; the raw token of the file (ID_OF_THIS_FILE, DATA_OF_EACH_LEVEL, MATERIALS_MILDSTEEL, DROP_CAP/PANEL, COMB_3) is in the tooltip. A block with no rows shows " holds no data rows."

The **Table** and **Text** tabs are hidden by default. So is the **Table** group of the View tab (**Add Row**, **Delete Row**, **Add Element Block**, **Delete Element Block**, which show only while the Table view tab is active). With the tabs hidden, a model opens on **Plan**; it is edited through the Plan and the dialogs. To show them again, switch on **Show INP Table/Text tabs (advanced)** (a small toggle in the View tab; saved in `plan.ini`, off by default).

Mouse and keyboard in the Plan

The left mouse button starts in **Window Selection/Pick mode** (**Model ▸ Selection ▸ Window Selection/Pick Mode**): a click selects, a left drag draws a selection window (see "Selecting items"). This is the home mode: it is active at start, after New, Open or Import, after a tab or view switch, after **Esc** and whenever Pan or Rotate is switched off. **Pan** (**Model ▸ Selection ▸ Pan**) makes a left drag pan the view. **Rotate** orbits the 3D view. They are radio-style: pressing the lit button returns to the home mode.

Wheel = smooth zoom about the cursor (touchpads too); **middle drag** = pan, also during a command; middle double-click = Zoom Extents; **Ctrl+click** = add an item to the selection, or remove it when it is selected; right click = context menu; hover = the item under the cursor is outlined and, after a short rest, labelled with its type and number. **Esc** in Pick mode clears the selection or cancels a window; in Pan or Rotate it returns to Pick. The cursor is a crosshair in drawing commands, an open hand in Pan mode and a closed hand while panning.

There is no floating navigation overlay on the canvas any more: use **View ▸ Zoom**, the wheel and the keys (F = Zoom Extents, T = top view, I = isometric view in 3D).

The Plan draws an adaptive background grid (lines at 1, 2 or 5 times a power of ten), switched with **View ▸ Display ▸ Look ▾ ▸ Grid** (a row of the Look list; it is not in the Layers list). This background

grid is only a drawing aid. It is not the named grid lines of the model (see "Grid lines"). It is off by default while the structural grid lines show.

The legend card at the top left of the Plan opens and closes with a click on its head. Until you click it, it starts closed when it would cover more than 15 % of the view; after a click OpenPT keeps your choice between sessions.

Zoom and camera commands

View ▸ Zoom acts on the view of the active tab: *Zoom Extents* (F) fits the drawing and leaves room so that the colour bar of a result sits beside the slab, never on it; *Zoom In* / *Zoom Out* make the view 25 % larger / 20 % smaller; *Redraw* draws the Plan and 3D views again. The floating zoom control of earlier versions is gone.

View ▸ Camera has *Decrease Z Scale*, *Reset Z Scale* and *Increase Z Scale*. They change the vertical exaggeration of the 3D view in steps of 0.5, between 1 and 10. This is for display only; the model is not changed. The 3D view shows "Z ×n" when the factor is not 1. The camera presets (Top, Bottom, Left, Right, Rear, Front and the isometric views) have no ribbon button: use the keys **T** (top) and **I** (isometric) in the 3D view, and orbit with **Rotate** or Shift + middle drag.

1.3 The 3D view

The **3D** tab shows the level chosen in the level combo, with a parallel (not perspective) projection. The default look is calm and **Shaded** (**View ▸ Display ▸ Shaded**; switching it off gives the wireframe look, and a user who switched shading off keeps it off):

- the slab top is one flat light-grey surface with openings cut out; a thicker slab region or a drop cap shows only below the slab, as a block (visible from below, through openings and at the free edge);
- beams as boxes hanging from the slab top (with their end offsets);
- columns and walls in grey (columns concrete grey): those stored "below" the slab go down to the plane below, those "above" go up to the plane above; circular columns are 16-sided prisms;
- tendons as 3D polylines along their profile, with joint dots, live-end arrowheads and dead ends drawn as a short bar across the tendon (a one-point tendon or an end-on view keeps a small circle). **View ▸ Display ▸ Look ▾ ▸ Group Colours** colours them by tendon group;
- loads as short arrows standing on the slab with a thin outline of the area; an area (patch) load has a closed lid joining the tails of its corner arrows and rows of grid arrows, so that it reads as one load; labels have a white halo;
- rebar when shown; when a result is shown, the coloured deformed shape (see "Results in 3D").

The 3D view follows the open ribbon tab like the Plan does (see "Tab Focus").

Mouse in the 3D tab: wheel = zoom about the cursor; middle drag = pan; **Shift + middle drag** = orbit; **Rotate** mode (**Model ▸ Selection ▸ Rotate**) orbits with the left button (Ctrl locks one axis); middle double-click = Zoom Extents; keys F, T, I = fit, top, isometric. In the home mode a left drag draws a selection rectangle (solid, left to right = whole item inside; dashed, right to left = crossing; Ctrl or Shift adds; Esc cancels; a short drag is a click). The selection is shared with the Plan, the Properties palette and the Modify commands. A left click selects the nearest beam, column, wall, slab or tendon edge (within 6 px). **Ctrl+click** adds an item to the selection or removes it. A click on empty space clears the selection.

You can also **model in the 3D tab**: the draw and modify commands work there on the working plane (the slab top of the shown level). If the plane is seen edge-on, the command bar says "Rotate the view to see the working plane (the plane is edge-on)" and clicks add no point. **Select by Type** also works in 3D (see "Select by Type").

1.4 Tab Focus

On the Plan and in 3D, items outside the family of the open ribbon tab are faded to one third: the **Tendons & Rebar** tab shows tendons and rebar strong and **Design** shows strips and design sections (on the **Loads** tab the tendons are the faded family). Slab, beams, columns and walls and results are never faded, and a selected item is never faded. Labels under 7 px are not drawn. **Loads are drawn only on the tab where they are worked on.** On the **Model** tab and on every other tab except Loads and Beam, no load symbol and no load value is drawn (the Beam tab keeps them: its Loads group edits beam loads); a load hidden this way cannot be picked either: a click, a selection window and **Select by Type** skip it, so a click on a beam that carries a line load selects the beam. On the **Loads** tab an **Area loads** key at the top left of the Plan names the value of each area load by its colour. The loads are drawn calmly: one small hexagon (12 px) at the centre of each loaded area, and the value labels, which go through the same label pass as the other plan labels (no overlaps; a label that finds no room is not drawn). With Tab Focus off the loads show on every tab. **View ▸ Display ▸ Look ▾ ▸ Tab Focus** turns it off or on (on by default; saved in `plan.ini`). While Tab Focus hides the loads on the open tab, the **Loads** button of **View ▸ Layers** stays pressed but is dimmed, and its tooltip says "Loads are drawn on the Loads tab." (for an OpenPT model, "Loads are drawn on the Loads and Beam tabs."). A selected load is cleared when its tab hides the loads. Reports, exports and the picture of a result are not affected.

1.5 Start view and what is drawn

Start view. Opening or importing a model shows only the **structural plan**: outlines of slabs, openings, drop caps, beams, columns and walls and the grid lines, each with its dimension text. Tendons, loads, rebar, support lines, splitters, tributaries and design sections are off, and their toggles in **View ▸ Layers** are off accordingly (the rebar is faded with no callouts on the other tabs unless you turned rebar on). Opening a family's ribbon tab shows that family (Tab Focus); drawing a tendon, load or rebar turns its layer on. **View ▸ Display ▸ Default Display** brings the structural plan back, unless you pressed **Save as Default** in View Settings (then it brings back your saved set). The start view applies to the window only; pictures and reports make their own choice.

Drawing standard. The plan reads like an engineer's drawing. Grid lines with circled labels (X: 1, 2, 3 at the top; Y: A, B, C at the left, without I and O) are derived from the columns and walls when the model has no grid of its own (not written to any file; at most 60 per direction); **Grid Systems...** makes them editable. Columns are filled dark at true size, walls filled, tendons drawn as fine lines (dark red along X, dark blue along Y, others dark grey). Where two neighbouring grid bubbles of one side would overlap, the second is moved outward (see "How grid lines are drawn"). Labels (tendon heights and tags, rebar callouts, load values on the Loads tab, item labels, strip names) never overlap: each tries its own place, then nearby places with a thin leader; a label that finds no room is not drawn (its tooltip still shows it). Load labels read value and unit with three significant figures: `Fz = 14.2 kN`, `w = 12.0 kN/m`, `q = 5.00 kPa`, `Mx = 3.50 kN·m`; grid dimensions read `8.40 m`. Units follow the display units in every view and picture.

View Settings. View ▸ Display ▸ View Settings opens the dialog **Select/Set View Items**. It has three tabs:

- **Structural Components** (slab region, column, wall, beam, drop cap/panel, opening, gridlines, rebar, support line, tributary region, splitter, ...);
- **Finite Element** (node, frame, shell, tendon, supports, springs, manual design section, ...);
- **Loads** (point, line and patch loads, plus a list of **Load cases** with check boxes, and an **Auto** box for the load symbol size).

Each row has the columns **Display** (draw the outline), **Render** (fill), **ID** (number), **Dimension** (size or value, for example "300 x 600"), **Label**, **Symbol**, **Symbol Size** and **Font Height** (both in model length units). Under each column, **All** / **None** buttons tick or clear the whole column; under the two size columns, type a number and click **All** to copy it to every row. For support lines, tributaries and splitters the Display column has separate **X** and **Y** boxes. On the **Gridlines** row only **Display** and the colour act: **Display** is the same switch as **Grid Lines**, and the colour is set in **Colour Palette** (see "Grid lines").

OK applies and redraws (view settings are not undoable and not saved in the INP); **Cancel** discards; **Save as Default** saves the state for new models and for **Default Display**; **Show by Type...** and **Colour Palette...** (View ▸ Display) set which kinds show and the colours per item kind (*Set White/Black Background and Update Colours*, *Clear Changes*). A colour set in the palette always wins over the default colours.

Layers of the View tab (**View ▸ Layers**). Beside the drop-down **Layers ▾**, the group has five small toggles, in this order: **Components**, **Tendons**, **Rebar**, **Loads** and **Mesh**. **Components** (tooltip title "Structural Components") shows or hides slabs (with their openings and drop panels), walls, columns and beams at once; it reads as on while any of them is shown, and the Layers list keeps one row per part. **Mesh** is for an OpenPT model only. While Tab Focus hides the loads on the open tab, **Loads** stays pressed but is dimmed (see "Tab Focus"). The adaptive background grid is not in this row: it is a row of **Look ▾** (below).

The **Layers ▾** list shows every layer with its check mark (a change made anywhere else shows here too). Its rows are nouns that name the layer; the buttons on other tabs keep their own captions (**Loads ▸ Show Loads ▸ Show Loads**, **Tendons & Rebar ▸ Visibility ▸ Show Tendons**). The list ends with two commands: **All Layers** (every layer on; the settings of the look, Tab Focus, Show Details and Group Colours, and Tendon Heights stay as they are) and **Model Only** (slabs, walls, columns and beams on; tendons, labels, loads, mesh, drawing and rebar off). The list holds:

- *Labels*, *Drawing* (drawn entities), *Grid Lines*, *Grid Dimensions* (the grid lines of the model; see "Grid lines");
- *Slab Regions*, *Walls*, *Columns*, *Beams*;
- *Tendons*, *Tendon Heights* (at each span's high and low points the height of the tendon in mm, and at each live end an anchor tag with the tendon name and count, for example `T7 x 4`; automatic while the Tendons & Rebar tab is open and in report tendon pictures, until you set it yourself), *Loads*, *Rebar* (twins of the large toggles of the Tendons & Rebar and Loads tabs, whose buttons read **Show Tendons** and **Show Loads**; Tendons, Rebar and Loads are also toggles of the group), *Mesh* (draws the mesh cell edges on the Plan and in 3D).

Look ▾ (**View ▸ Display ▸ Look ▾**) holds the settings of how the views are drawn that are not layers:

- *Grid* (the adaptive background grid of the Plan);

- *Tab Focus* (above) and *Show Details*: while a contour, a diagram or a deformed shape shows, rebar, loads, labels and strips are not drawn and tendons are drawn at 25 % in grey; **Show Details** (off by default) brings them back;
- *Group Colours* (colours the tendons by group).

In **View** ▸ **Display**, **Settings** (View Settings), **Show by Type...** and **Colour Palette...** are large buttons (icon above the word), **Look** ▾ is a drop-down, and **Shaded** and **Default Display** are small buttons.

The **View** ▸ **Section** group has *Section Cuts* (show or hide) and *Create Section Cut* (see "Section Cut").

1.6 Properties palette

Model ▸ **Edit** ▸ **Properties Palette** (Ctrl+Shift+P, alias PR or PROPERTIES) shows or hides the docked Properties pane. It follows the selection. Its General rows include **Layer** and **Colour** (see "Layers"). The visible state is remembered (default shown). The pane edits the same fields as the **Item Properties** window (see "Item Properties").

1.7 Interface language

OpenPT has two interface languages: English (the default) and Vietnamese (**Tiếng Việt**).

Where. The two items **English** and **Tiếng Việt** are at the end of the **Help** ▾ menu, check-marked for the language that runs. Each is written in its own language, whichever language is running.

When it changes. At once, without a restart. The open models, the selection, the undo stack, the active tab, the window sizes and the zoom are kept. A status line that is already written keeps its language until the program writes a new one.

Where it is saved. In `%APPDATA%\OpenPT\plan.ini`, key `Language`, value `en` or `vi`. Any other value is read as English.

Command line. `--lang en` or `--lang vi` stands anywhere on the command line (see "Command-line use"). It sets the language of that run only and does not change `plan.ini`. A run that starts with a switch (pictures, conversions) does not read `plan.ini` and uses English unless `--lang vi` is given. For any other value OpenPT writes "Unknown --lang value <value> (en or vi); English is used." and uses English.

What is translated. In Vietnamese the text that is fixed in the program is drawn in Vietnamese: ribbon tabs, group names and buttons, tooltips, menus, context menus, dialog titles, labels and buttons, the hint of the Point: box, the strip design texts (the Designed line, the design dialog text blocks, the criterion codes STRENGTH, INITIAL ... and the Messages column), and the names of the beam result leaves. The Word calculation report is written in the program's language.

What stays English. Text that OpenPT builds while it runs and that is not in the dictionary (some messages with numbers or names); the Windows message boxes and file dialogs; the names of files, levels and items; the files OpenPT writes (INP, .adm, .optp, CSV, PDF, spreadsheets, IFC, DXF); the engine texts of some design warnings; the text of the Report Compiler (English); and the text the command-line switches print.

The dictionary file. The Vietnamese texts are in `lang\vi.txt`, in the `lang` folder beside `OpenPT.Gui.exe`. OpenPT reads it once, when it switches to Vietnamese; English never reads it. You may edit it (UTF-8, in a text editor) and switch the language again.

- One entry per line: `English text = Vietnamese text`. The separator is the first " = " (space, equals sign, space) of the line.
- A line that starts with `#` is a comment. Empty lines are ignored.
- In both halves `\n` stands for a line break, `\\` for a backslash and `\=` for an equals sign. The spaces around each half are dropped.
- The lookup uses the whole English text, with capital letters as written, and ignores the underscore of an access key. A text that ends in `...`, …, `:` or `▾` is looked up again without that ending. A text with several lines that has no entry as a whole is looked up line by line.
- Where there is no entry, the English text is drawn. If a later line has the same English text as an earlier one, the later line is used.
- A line without the separator, or with an empty half, is skipped. If the file is missing or cannot be read, the interface stays English.

1.8 Help ▾

The **Help** ▾ menu at the right end of the tab row has:

- **Documentation...** opens the window "OpenPT Documentation" with the tree *License Agreement and Copyrights*, *Manuals* ▸ *OpenPT User Manual*, *User Guide* ▸ *OpenPT User Guide* and *Verification* ▸ *Verification Manual* (30 illustrated cases) and *Verification Manual - full* (188 cases). Double-click an entry, or select it and click **Open**, to open its PDF in your default PDF viewer. The five PDFs are in the `docs\` folder beside the program file.
- **Associate .optp files with OpenPT** (see "File association").
- **About OpenPT**.
- **Licence...** shows the licence state. A build that does not enforce a licence says "This build does not require a licence". In a ship build, after the trial ends or without a valid licence the program does not run: the start shows the licence dialog in locked mode ("OpenPT needs a valid licence to run.", button Exit); a licence that becomes invalid while the program runs disables the window and the title reads "OpenPT - licence required". Command-line runs without a valid licence exit with code 3. One installer belongs to one personal download; an installation is bound to the computer and the account. The command **Time-Dependent Analysis...** needs the licence feature "analysis".
- **Keyboard Shortcuts...** opens a window to view and customise shortcut keys; each row shows the command, where it stands on the ribbon, and its key. Custom bindings, and the command aliases you give, are saved in `%APPDATA%\OpenPT\shortcuts.txt`.
- **English** and **Tiếng Việt** (see "Interface language").

1.9 Display units

The **Units** drop-down at the right end of the status bar has **SI** (default), **US** and **Consistent units...**.

- **SI** shows m, mm, kN, kN·m, MPa. **US** shows ft, in, kip, kip·ft.
- **Consistent units...** opens a dialog with a **Force** and a **Length** combo; every other unit follows from the two.

The setting applies to the cursor coordinates, the dimensions on the plan, labels and report pictures, and to the dialogs that have been converted to display units. Typed coordinates follow the plan length unit (in SI, `6,0` is 6 m). The units are saved with the model (only when not SI) and are not an undo step. The **File** ▸ **New model...** dialog has a **Units** ▾ button at its bottom left.

1.10 Long runs, stability and recovery

Long runs. Every long run (mesh, analysis, design, punching, cracked deflection, modal, SAFE-type jobs, saving and opening a project) shows its progress in the status bar: a bar of 180 × 12 pixels, a text "step - NN % - m:ss", and a **Cancel** button when the run can be stopped (tooltip "Stops the run."); the Windows taskbar button shows the progress too. The percentage never goes backwards; a step that cannot be measured shows an indeterminate bar. **Esc** does the same as Cancel. A cancelled run changes nothing and the status says "Cancelled." Opening, creating or importing a model cancels a running run. Window commands run in the background where they are heavy: opening and saving a project (with Cancel), Generate Rebar, Generate Sections, and imports; the window never freezes for them.

Safety net. Every ribbon and menu command runs under a guard. If an unexpected error occurs, one message box shows, for example "{command} failed: ... The model may have changed: check it, or use Undo. Details are in <log>."; a half-made undo step is rolled back and the error is written to `error.log`. Repeats within 10 seconds are logged once. If errors keep coming, a last box says "OpenPT keeps failing in the same place ... Save the model under a new name and start OpenPT again." A view that cannot be drawn shows "This view could not be drawn: <message>". Start-up failure shows "OpenPT could not start"; closing may ask "OpenPT could not finish closing ... Close anyway?". If the window hangs for more than 5 seconds, a line is written to `hang.log`. For unhandled errors OpenPT also writes a local diagnostic archive `crash_<timestamp>.zip` under `%APPDATA%\OpenPT\crashes\` (the 20 newest are kept) and shows a dialog with **Copy Details** and **Open Crash Folder**. Nothing is uploaded automatically. Send the log files with your report.

Recovery copy. Every 3 minutes, when the document has changed and nothing is running, OpenPT writes a recovery copy (`plan.ini` key `RecoveryMinutes`; 0 = off). After a crash, the next start asks "OpenPT did not close normally last time. A recovery copy of <name> from <time> exists. Open it?". **Yes** opens it as "<name> (recovered)" (untitled: Save asks for a name); **No** keeps it for later; **Delete** removes it.

Memory. Before a big allocation (factorisation, analysis, mesh) OpenPT checks the free memory and refuses with "Not enough memory for <what>: it needs about X GB and Y GB is free. ..." instead of crashing. A run is stopped when the PC runs out of memory ("The run was stopped: the PC was running out of memory ..."). When OpenPT's own memory passes 70 % of the PC's memory, its caches are emptied and the info line says "Memory is high: ... new runs are refused until memory is freed. Save your work, close other programs, or restart OpenPT." Setting the environment variable `OPENPT_MEMORY_GUARD=0` turns these checks off.

Messages. Every message says what happened and what to do, in British spelling, with level names and "Tendon 7" in place of internal identifiers such as `LEVEL_2` and `TENDON_7`. The files OpenPT writes keep the identifiers.

1.11 Dialogs

Every dialog ends with **Apply** (where it has one), **OK** and **Cancel** at the bottom right; OK is the default button and **Esc** cancels. A dialog that only shows information has **Close**. The Tendon Editor works the same way (OK = apply and close).

1.12 Results display basics

After an analysis, the result opens in the Plan and 3D views and the **Results View** pane at the right of the window. The chapter "Results" describes it in full: the Load drop-down, the Analysis and Display tabs, the colour bar, bands, labels, max and min markers, strip moment diagrams and the 3D result.

1.13 Keyboard Shortcuts

Open **Help** ▾ ▸ **Keyboard Shortcuts...** to view and customise shortcut keys (see "Help ▾"). The keys named in this manual: Ctrl+N, Ctrl+O, Ctrl+S, Ctrl+Z, Ctrl+Y, Ctrl+F1 (ribbon), F9 (Run Analysis), F5 (Check Model), F4 (report panel), F (zoom extents), F3 (snap on/off), F7 (grid snap), F8 (ortho), Ctrl+1 to Ctrl+7 (snap modes), Ctrl+Shift+P (Properties palette).

2 Files and projects

2.1 File types

OpenPT has one native file, the **OpenPT project** `.optp`. Every other model file is imported (read into a new OpenPT model) or exported (written from the open model).

File	Open	Import	Save	Export	Notes
OpenPT project `.optp`	**Open**, drop, double-click	–	**Save**, **Save As**	–	One file holding the model, its sidecars and its results.
ADAPT exchange file `.INP`	–	Import model...	–	ADAPT model (.INP)...	The model format of ADAPT-Builder 2019 (INP version 3 or 4).
ADAPT-Builder project `.adm`	–	Import model...	–	ADAPT-Builder model (.adm)...	Single-floor models (three planes).
SAFE 12 / 20 / 22 text file `.F2K` / `.$sf`	–	Import model...	–	SAFE model (.F2K)...	The export writes the SAFE 20 and later table format, or SAFE 12.
SAFE database `.FDB`	–	Import model...	–	SAFE model (.FDB)...	Reads SAFE 12, 16, 20 and 22; the export writes SAFE 22.
IFC file `.ifc`	–	–	–	Model to IFC (Revit, Tekla, viewers)...	Every level as solids (IFC2X3).
Revit exchange file `.optrvt.json`	–	–	–	Model to Revit (native)...	Read by the OpenPT add-in for Revit 2024 and later.
DXF / DWG `.dxf` / `.dwg`	–	Import DXF/DWG...	–	Structural plan to AutoCAD (DXF)...	Import brings CAD geometry in as drawn items; export writes the plan of one or all levels.
Word report `.docx`, CSV files	–	–	–	Calculation Report, the CSV exports	See "Exporting".

The **Open** dialog (**File** ▸ **Open...**, Ctrl+O; also on the quick access toolbar) has the filter *OpenPT project (*.optp)* and *All files*. A model file of any other kind that you choose there, drop on the window, give on the command line or click in **Recent files** is imported instead (next section). If the open document has unsaved changes, OpenPT asks "Save changes to <name>?" (Yes / No / Cancel) before it opens another file.

2.2 Creating a new model

1. Click **File** ▸ **New model...** (Ctrl+N).
2. Fill in the **New model** dialog (the fields are in SI units; the **Units** ▾ button at its bottom left changes them):

Defaults: storey height H 3000 mm (Level 2 at H, Level 3 at 2H); concrete f'c 35 MPa, unit weight 24.5 kN/m³, label C35; rebar fy 500 MPa, label Rebar 500; strand Ep 195000, fpu 1860, fpy 1674 MPa; *Planned file path* (optional, written into the file's ID lines; the first Save still asks for a name).

The dialog has no design-code row: a new model takes the **design code last used** (EC2 the first time). Change it afterwards with **Design ▸ Criteria ▸ Design Code...** (see the chapter "Loads, criteria and combinations"). 3. Click OK. The new model has three planes (bottom, current, top) and is untitled until the first Save.

2.3 Importing a model

File ▸ Import ▸ Import model... is the one command that reads a model made by another program. It reads **INP**, **ADM**, **F2K**, **$sf** (SAFE 22 table file) and **FDB** (SAFE 12, 16, 20 and 22 databases) files; it looks at the content of the file, not at its extension. The result is always a new **untitled** OpenPT document, marked as changed, with no link to the source file; the source file is never written. Save it as an `.optp` to keep it.

- INP and ADM files are read as they are (see below).
- SAFE formats are converted with the default converter options and **no dialog**. The import report goes to the info bar and to the Check Model window (**Analyse ▸ Check ▸ Check Model**) and, for a model made from a SAFE file, to the **Report** tab of the Results View pane. A file `<name>.import.txt` next to the source file lists what is not held (written when the folder can be written).
- Dropping a file on the window, giving it on the command line, or clicking it in **Recent files** does the same.
- **Import DXF/DWG...** (File ▸ Import) brings the lines, polylines, circles, arcs, text and dimensions of a CAD file into the open model as drawn items (see "Draw tools"). **Model ▸ Selection ▸ Calibrate** appears only while the model holds imported drawing items; it scales them to a known distance.

What a SAFE import produces, and how to work with it, is in the chapter "Working with SAFE models". The older commands "Design strips from ADAPT (.adm)..." and "SAFE F2K to OpenPT model..." no longer exist: the first import of an .adm brings in its design strips, and the second is Import model.

ADAPT exchange files (.INP)

OpenPT imports any ADAPT-Builder 2019 exchange file (INP version 3 or 4) and shows all of its content in the model tree (the Table and Text tabs show it too, once you switch them on: see "Model tree and the Table and Text tabs").

- **Check Model** (F5, **Analyse ▸ Check ▸ Check Model**) checks the model against the rules of the INP format (for example lines longer than ADAPT can read, characters that cannot be stored) and lists the problems in the **Check Model** window (a summary and the list of messages; double-click a message to jump to its line or item).
- OpenPT writes INP files that ADAPT-Builder can import (**Export ▸ ADAPT model (.INP)...**). A model with more than three planes (see "Multi-level models") is written in the layout ADAPT uses for its own multi-level exports.

ADAPT-Builder project files (.adm)

1. Click **File** ▸ **Import** ▸ **Import model...** and choose the `.adm` file.
2. OpenPT reads the model directly from the .adm. When it succeeds the info bar says "Imported <name>.adm: read directly from <name>.adm, identical to the saved model."
3. The design strips stored in the .adm (support lines, splitters, tributaries, design sections) are imported into the level.

The document is untitled: OpenPT never writes into ADAPT's project folders on its own.

Fallback to ADAPT's INP copy. The direct reader supports ADAPT-Builder 2019 files of a single floor (three planes) in N-mm units. If it cannot read a file (other version, other plane count, damaged file), OpenPT uses the INP copy that ADAPT-Builder writes when auto save is on (`<project>\<name>\backup\<name>_NNN.INP`) or an INP exported next to the .adm, and adds "(Direct read of the .adm failed: <reason>; used ADAPT's INP copy.)" to the message. If there is no copy, OpenPT says "No INP copy of <name> was found." and asks you to switch on *Auto Save Setting* > *Enable auto save and backup* in ADAPT-Builder or to export the model to INP.

What is not read from the .adm: the column and wall reactions that ADAPT stores (the REACTIONS block is left empty; OpenPT computes its own).

2.4 Saving: .optp only

File ▸ **Save** (Ctrl+S) and **File** ▸ **Save As...** write an `.optp` and nothing else. The first Save of an untitled model asks for a name. **Results are always included when the model has results**: there is no Yes/No question. Save lists what the project does not carry ("Not saved in the project: ..."). A save runs in the background with a Cancel button (Esc too): a save that is cancelled leaves the file on disk unchanged, the document stays changed, and the message says "Saving was cancelled; the file on disk is unchanged." A save cannot be edited under, and closing the window during a save waits for it.

To write an INP, .adm, F2K or FDB file, use **File** ▸ **Export** (next sections).

2.5 Sidecar files

OpenPT keeps data that the INP format cannot hold in small JSON files beside the model (the "sidecars"). In a project they travel inside the `.optp`; an export packs them with the loose files only where the target format has a place for them.

Sidecar	Content
`<model>.inp.strips.json`	Strips (support lines, splitters, tributaries, design sections), section cuts, drawings, per level; the grid systems of the whole model (key `gridSystems`); the base rebar of a model made from a SAFE file (key `baseRebar`)
`<model>.inp.criteria.json`, `.combos.json`	OpenPT criteria and options; load combinations
`<model>.inp.openpt.json`	Data of a model made from a SAFE file that the INP cannot hold (springs, imposed displacements, modal and spectrum cases)
`<model>.beamdesign.json`	Beam design options, T-section flanges, optional given bars

Sidecar	Content
`<model>.etabs_forces.json`	ETABS forces kept for the strip design (see "Sending a floor to ETABS")
Layers file	Layers, objects' colour, display units (key `displayUnits`, only when not SI)
`<name>.import.txt`, `<file>.export.txt`	Import and export reports

Drawn entities (**Model ▸ Draw**) live **only** in the strips sidecar: an INP that goes through ADAPT-Builder and back does not carry them. If a sidecar cannot be read, OpenPT leaves it alone and opens the model without it.

2.6 Results store

A project saves the analysis results of the model, so that reopening it gives the results at once, without a new analysis. Inside the project the results store has this layout:

```
<stem>_OpenPT\
    openpt-results.json              manifest
    cracked.json                     cracked deflection (when run)
    Databases\<stem>ado\
        <Case>.ADO                   text, readable, one per load case
        <Case>_TOTAL_DISPLACEMENTS.bin, <Case>_REACTIONS.bin, <Case>_LOADS.bin
        MODES.bin                    periods, participation, mode shapes (when present)
```

When it is written. On every successful Save of a model that has current results (results exist, mesh current, every load case solved). Design-section and punching runs, the cracked deflection, modal results and the strip design are recorded too. A failure to write the store never fails the save; a red message "The analysis results could not be saved: <message>" appears. **Analyse ▸ Run ▸ Save** (Save results; F2K-kind documents) writes the store again by hand. The command **Delete saved results**, which deletes the saved store so that the next Save packs the project without results, has no ribbon button in this build.

When it is read. On opening, OpenPT checks that the store belongs to exactly this model (a fingerprint of the model text and of each sidecar), re-meshes with the stored settings and checks that the mesh is identical. If all matches: "Loaded the saved analysis results of <date> (<n> load cases, Level <k>)." (design, punching and cracked results too), the **strip design is restored from the store instead of being computed again** (much less CPU and memory at opening), and the model opens **locked**. Otherwise nothing is loaded and the message says why: the model changed after the save ("... are out of date ...; run the analysis again."), the mesh differs ("... do not match the mesh ..."), or the strips or options changed ("Saved design results are out of date ..."). Saved cracked results made with load combinations that are different now are reported out of date. Opening a project with results and saving it no longer freezes the window: both run in the background under "Opening..." / "Saving {file}..." with a Cancel button.

A model made from a SAFE file with imposed displacements, whose saved results were solved with them applied, shows the results as not current: "Saved analysis results of <date> were solved with the imposed displacements of the SAFE file applied; run the analysis again." (see "Working with SAFE models").

2.7 Locking the model

As in SAFE, a model with analysis results is **locked**, so that the results always belong to the model on screen.

- The **Lock/Unlock Model** button (padlock) is in **Analyse ▸ Run ▸ Lock/Unlock Model**, next to Run Analysis. It shows an open grey padlock while the model is unlocked and a closed padlock with an amber (orange) body while it is locked; the label and the tooltip do not change.
- The model locks itself when an analysis finishes (single-level, multi-level or SAFE-type), and when a project is opened with a current results store. You can also lock a model by hand (edits are then refused even without results). The status bar and the Lock/Unlock command never wait for a running solve.
- While locked, the title shows a lock mark and every command that changes the geometry, loads or supports is refused with "The model is locked. Unlock it to edit (this deletes the analysis results)." This covers geometry, properties, loads, tendons, rebar, criteria, mesh generation, Story Manager, Copy, Vertical and Undo/Redo of model edits.
- Still allowed while locked: viewing, results, tables, export, design of sections from the present results, punching check, cracked deflection, editing load combinations (the dependent design results become out of date), grid lines, and the **strip modelling**: support lines, splitters, tributaries and design sections can be created and edited while the model is locked, without asking to delete the results; the design becomes out of date and the status bar says **Design out of date**.

To unlock:

1. Click the padlock.
2. OpenPT asks "Unlocking the model will delete all analysis results (single-level, multi-level, design, punching, cracked deflection). Continue?"
3. **Yes** deletes all results in memory and the results store of this model, clears the contours and diagrams and allows editing. **No** changes nothing.

If a model is unlocked while results are present, the first edit asks the same question.

2.8 Single-file projects (.optp)

An `.optp` file is one file (a ZIP archive) that holds the model, all its sidecars and, when there are results, the results store. Use it to archive or send a complete job.

To save a project: File ▸ Save As..., choose the file name. **To open a project: File ▸ Open...**, or drag the `.optp` onto the OpenPT window, or double-click it in Explorer (after the association below), or start `OpenPT.Gui.exe <file.optp>`. OpenPT checks every entry of the archive, extracts it to a working folder `%LOCALAPPDATA%\OpenPT\projects\<name>_<code>\` and opens the model there. The recent-files list holds the `.optp` path.

Working with a project:

- **Save** (Ctrl+S) packs the working folder into the `.optp` again (through a temporary file, so a failed or cancelled save never damages the old file).
- After an analysis the results go into the working folder; the `.optp` gets them at the next Save, and the title shows `*` until then.

- The working folder is deleted when the document closes. Working folders older than 7 days that no open document uses are removed when OpenPT starts.
- The command line `--pack-optp` and `--unpack-optp` pack and unpack projects (see "Command-line use").

A model that you imported is untitled until you Save As; if OpenPT closes unexpectedly, the recovery copy keeps your work (see "Long runs, stability and recovery").

2.9 Exporting

File ▸ Export writes a copy of the open model or of its results. **Every export only reads the document**: the open document, its file and its lock are untouched, the changed mark does not change, and the export works while the model is **locked**. Each model export ends with a report of what was written and what the target program will not get (for example drop caps, springs).

Command	Writes
ADAPT model (.INP)...	An ADAPT exchange file.
ADAPT-Builder model (.adm)...	An ADAPT-Builder project (single floor, three planes).
SAFE model (.F2K)...	A SAFE text file.
SAFE model (.FDB)...	A SAFE 22 database.
Model to IFC (Revit, Tekla, viewers)...	Every level as IFC solids.
Model to Revit (native)...	An exchange file for the OpenPT add-in of Revit.
Structural plan to AutoCAD (DXF)...	The plan of one level or of several levels.
Export Displacements (CSV)...	The displacements of the last analysis (disabled until a run exists).
Export step..., **Export all...**	CSV files in the layout of SAFE's tables: one step, or every case, mode, spectrum case and combination.
Export Design Table (CSV)..., **Export Cracked Table...**	The strip design and the cracked-deflection sections.
Calculation Report	A Word `.docx` of the slab (see "Calculation Report (.docx)" in the chapter "Design").

Report Compiler output (RTF and PDF) is made from **Report ▸ Compiled Reports**.

Export to ADAPT: INP and .adm

ADAPT model (.INP)... (**File ▸ Export ▸ ADAPT model (.INP)...**) asks for a file name and writes the model. For a model that has more than three planes, the INP is written in ADAPT's own multi-level layout; ADAPT-Builder imports such a file ("Create new model", all levels).

ADAPT-Builder model (.adm)... (**File ▸ Export ▸ ADAPT-Builder model (.adm)...**): choose the file name. The .adm holds the levels, materials, sections, beams, columns, walls, slab regions, openings, loads, load cases, combinations, groups, tendons, base and mesh rebar; the FE mesh when it belongs to the present model; the analysis state and deflection contour when the results are current; and the support lines, splitters, tributaries, design sections and calculated rebar of the meshed (or shown) level.

With current results OpenPT also writes the **result files** ADAPT reads when you display a result: `<folder>\<stem>\Databases\<stem>ado\*.ADO` and `*.bin` per analysed case and combination, plus

`cracked_<combination>.ADO`. Before that it deletes only the old `*.ADO`, `*.ado`, `*.bin` there and `<stem>.adi`, `.dsi`, `.dso` in `Databases\`; everything else in ADAPT's folder is untouched. If this step fails the export still succeeds, with a red message "The analysis results could not be written for ADAPT: <message>". The .adm writer writes one result set at a time to temporary names.

What is not written, and refusals. Before writing, OpenPT asks "The .adm will not contain: <list>. ADAPT will open the model without them. Save anyway?" (default No); the final message adds "Not in the .adm yet: ...". Possible items: support lines of other levels; release and stiffness-modifier rows; distributed bars without an extent row (written as single bars); design sections and calculated bars without current analysis; results without a current mesh; design criteria taken from defaults; imposed displacements, modal and spectrum cases and TCVN punching results; names with characters ADAPT cannot store (written with `?`).

OpenPT does not export an .adm when the model has more or fewer than three planes (it tells you to use the INP format instead; "This model has N planes. The ADAPT project (.adm) writer holds exactly 3 planes."), or when it holds generic sections or materials (generic beam shapes, column type 3), drop caps, ramps, no concrete material, or units other than N-mm (a "Cannot write <path>" message names the reason). Load case or combination names that contain `\ / : * ? " < > |` cannot be written. If anything goes wrong, the old file is left untouched (OpenPT writes to a temporary file, reads it back, and only then replaces the target). Models with drop caps therefore have to be exchanged with ADAPT through INP. Cracked deflection stays disabled in ADAPT-Builder for a written file.

Export to SAFE: .F2K and .FDB

SAFE model (.F2K)... writes a SAFE text file. The export form (shown for every interactive export) offers **SAFE version: SAFE 20 and later (table format)** (default) or **SAFE 12 (legacy)**. Columns, walls below and openings are written. An export from an OpenPT model writes **vertical offsets on**: every beam hangs from the slab top (SAFE keeps the T action, tendon eccentricities and soffit steps), the export report `<file>.export.txt` has the line "Vertical offsets: on (beams top-flush with the slab, slab soffits stepped)", and beam offsets other than top-flush are not carried (stated in the report). The report lists what the target will not get (for example drop caps, springs).

SAFE model (.FDB)... writes a SAFE 22 database (built from a template; objects OpenPT owns). Wrong or damaged files are refused with a message. Tendons, curved frames, ribbed or waffle slabs and steel sections are not read or written. OpenPT reads SAFE 20 `.FDB` files (format 19.09) on import; classes of a version the program cannot read are reported by name. Writing a SAFE 20 database is not offered.

> The installed SAFE 22 cannot import a SAFE 12 .F2K. Use the default table format for SAFE 20 and later.

2.10 Exporting to Revit, IFC and AutoCAD

OpenPT writes the model as a native exchange file for Revit, as a 3D IFC file, and as a plan DXF file for AutoCAD. The three commands only read the document: they write a new file and change nothing in the model, its sidecars or the changed mark. They work on every open document, and they work while the model is **locked**. They are greyed with no file open. **Tendons and rebar are written** in all three.

Command (File ▸ Export)	Writes
Model to Revit (native)...	An exchange file `.optrvt.json` for the OpenPT add-in of Revit
Model to IFC (Revit, Tekla, viewers)...	An IFC2X3 file of every level
Structural plan to AutoCAD (DXF)...	A DXF plan of the levels you choose

If the model is still being built or analysed, the command says so and writes nothing.

Model to Revit (native)

1. Open the model.
2. Click **File ▸ Export ▸ Model to Revit (native)...** and choose the file name. It writes a file `<name>.optrvt.json`.
3. In Revit 2024 or later, open the tab **OpenPT** and click **Import OpenPT model**, then choose the file. The add-in builds Levels, Floors (with real openings), Structural Columns, Framing, Walls, the tendons (as free-form rebar) and the rebar.
4. Importing the same model again asks "Replace the N elements of the previous import?" and replaces them.

Setup copies the add-in when Revit is installed; restart Revit once after the installation. The command line `--export-revit <model> <out.optrvt.json>` writes the same file (see "Command-line use").

Model to IFC (Revit, Tekla, viewers)

The file holds every level of the model as solids, in one file.

1. Open the model.
2. Click **File ▸ Export ▸ Model to IFC (Revit, Tekla, viewers)...**.
3. In the dialog, choose the folder and the name. The dialog starts in the folder of the document with the name `<document name>.ifc` (characters that a file name cannot hold become `_`). The file types are *IFC file (*.ifc)* and *All files*.
4. Click Save. The info bar shows the summary line.

If the file cannot be written, a message box and the red command bar text say "Cannot write <file>" with the reason, and no file is left behind.

What is in the file

- The format is IFC2X3 (ISO 10303-21 text, view definition "CoordinationView_V2.0"). The length unit is the millimetre. The X and Y of the model are written as they are: no scaling, no shift of the origin.
- The structure is Project (named after the document) ▸ Site ▸ Building ▸ one Building Storey for **every level** of the model. A storey has the label of the level and its elevation. A level without solids still gets its storey.
- Each element is an extruded prism: its plan polygon between a bottom and a top elevation, with the heights that the 3D tab uses.

OpenPT item	IFC element	Notes
Slab region	IfcSlab (floor)	Openings that lie wholly inside the slab region are cut out of it as voids.

OpenPT item	IFC element	Notes
Drop panel	IfcSlab (floor)	A slab of its own; its object type is "Drop".
Beam	IfcBeam	
Column	IfcColumn	Each column has the property set `Pset_ColumnCommon` with `LoadBearing` = true.
Wall	IfcWall	
Opening	A void of a slab region	An opening that does not lie wholly inside one slab region is not written.

- The name of an element is its label in OpenPT. An item without a label is named `Slab 1`, `Drop 2`, `Beam 3`, ... in order within its level.
- A column or wall that a level draws above its slab is the same member that the level above draws below its own slab. It is written once, with the level above.
- A face that is not level (a top or bottom that slopes) is written flat at its mean elevation. The summary line says how many were flattened.
- The element identifiers (GlobalId) are made from the document name, the kind of element, the level and its place in the list, so the same model gives the same identifiers each time.
- Text outside plain ASCII (for example Vietnamese letters in a label) is written with the escape codes of the IFC format.

Not in the file: loads, support lines and other strips, drawn entities, grid lines, materials and section properties (beyond the properties named above), and results.

The summary line has this form (the numbers are an example):

```
IFC: 3 levels, 12 slabs, 4 drops, 20 beams, 30 columns, 10 walls, 5 openings cut; skipped 2 (no thickness 1, opening not inside one slab 1); flattened 1 -> D:\Jobs\B01\B01.ifc
```

The `skipped` and `flattened` parts appear only when something was skipped or flattened. An item is skipped when it has no usable polygon, thickness, depth or height. Each reason is named with its count.

To use the file, open or link it with the IFC commands of the receiving program. OpenPT does not control what that program does with it.

Structural plan to AutoCAD (DXF)

1. Open the model.
2. Click **File ▸ Export ▸ Structural plan to AutoCAD (DXF)...**.
3. The dialog asks first: - **Levels: Current level** (default; the level that the level combo shows), **All levels**, or **Choose...** (pick the levels); - **One file with levels side by side** (default) or **One file per level**; - **Include** tendons, rebar and labels.
4. Choose the folder and the name. The dialog starts in the folder of the document with the name `<document name>_<level>_plan.dxf`. The file types are *DXF file (*.dxf)* and *All files*.
5. Click Save. The info bar shows the summary line.

What is in the file

- The format is DXF release 12 (ASCII text). The X and Y of the model are written as they are, at Z = 0, with no scaling and no shift of the origin. The file has no unit setting.
- Slabs, openings, columns, walls, beams and drop panels are closed 2D polylines of their plan polygons. Each tendon is an open polyline through its start point, swerve points and span ends. Tendons and rebar are written when they are ticked in the dialog.
- Each kind has its own layer and colour:

Layer	Content	Colour (AutoCAD index)	Linetype
`OPT-SLAB`	Slab regions	7 (white or black)	Continuous
`OPT-OPENING`	Openings	1 (red)	Continuous
`OPT-COLUMN`	Columns	2 (yellow)	Continuous
`OPT-COLUMN-ABOVE`	Columns drawn above the slab	2 (yellow)	Dashed
`OPT-WALL`	Walls	3 (green)	Continuous
`OPT-WALL-ABOVE`	Walls drawn above the slab	3 (green)	Dashed
`OPT-BEAM`	Beams	4 (cyan)	Dashed
`OPT-DROP`	Drop panels	6 (magenta)	Dashed
`OPT-TENDON`	Tendons	5 (blue)	Continuous
`OPT-TEXT`	All text	7 (white or black)	Continuous

- The layers you made in the model (**Model ▸ Selection ▸ Layer Settings**) go into the file with their colour and linetype; an off layer is written with a negative colour, and the lock flag is kept.
- Text is on layer `OPT-TEXT`, 250 high. A slab gets its label and `t=<thickness>` (a whole number) at its centre. A column gets its label beside it. A wall gets its label at the middle of its axis. A beam gets its label and its size `<width>x<depth>` along the beam. A drop panel gets its label at its centre. A tendon gets its label at its first point. An item without a label gets no label text (a slab still gets its thickness). Under the plan there is a title, 500 high: `<document name> - <level> - structural plan`. Text outside plain ASCII is written with the DXF escape `\U+hhhh`.

Not in the file: loads, support lines and other strips, section cuts, drawn entities (**Model ▸ Draw**), grid lines, results and dimensions.

The summary line has this form (the numbers are an example):

```
DXF: Level 2: 4 slabs, 6 openings, 52 columns, 8 walls, 30 beams, 2 drops, 90 tendons, 130 texts -> D:\Jobs\B01\B01_Level_2_plan.dxf
```

The `texts` count includes the title. When an item has no usable polygon (or a tendon has fewer than two distinct points), it is skipped and the line adds `; skipped n (reason count, ...)` before the `->`.

From the command line

`--export-ifc`, `--export-dxf` and `--export-revit` run the same exports without a window; see "Files and projects" in the chapter "Command-line use".

2.11 File association

OpenPT does not change the registry on its own. To open `.optp` files by double-clicking them:

1. Click **Help** ▾ ▸ **Associate .optp files with OpenPT**.
2. The association is written for the current Windows user only (HKCU). After that, double-clicking an `.optp` file starts OpenPT with it.

INP, .adm and SAFE files are not associated; import them from OpenPT, drop them on the window, or give the path on the command line.

3 Building and editing a model

All editing commands work on the **level shown** in the level combo. Every edit is one undo step (Undo Ctrl+Z, Redo Ctrl+Y, up to 100 steps). Edits are refused while the model is locked.

3.1 Adding structural components

The buttons are in **Model ▸ Add Structural Components**: **Slab**, **Wall**, **Column**, **Drop Panel**, **Beam**, **Support**, **Opening**. The **Support** tool is described in "Supports" below.

Component	Input	Finish
Column, Drop Panel	1 click: the centre	Each click places one item
Beam, Wall	2 clicks: start and end of the axis	Each second click places one item
Slab region, Opening	3 or more corner clicks	Enter, End, `C`, double-click, or right-click ▸ Close / End / Accept

Procedure:

1. Click the button. It stays checked while the command runs.
2. Optional: click **Item Properties** (**Model ▸ Modify ▸ Item Properties**) to set the size, offsets, material etc. of the next items.
3. Click the points in the Plan (or 3D) view, or type them in the **Point:** box.
4. The command repeats. Press **Esc**, type `x`, or right-click ▸ Exit to end it. Clicking the same button again also ends it.

Keys while a command runs:

Keys: **Esc** or `x` exit (unfinished points are discarded); **Enter**/**End** finish or close (beam/wall: end the chain); **C** close the polygon; **Backspace** remove the last point; **Ctrl+Z** remove the last point while points are pending, otherwise undo; **Shift** held = Ortho for this point.

Continuous Modeling. The arrow under **Wall** and **Beam** (and Line, Line Load) offers the check item *Continuous Modeling*: the end of one beam or wall becomes the start of the next. It is one setting shared by all these tools and is remembered.

While you draw, a rubber band runs from the last point to the cursor, with the length and angle next to the cursor (for example "3250 mm ∠ 90°"). Columns, drops, beams and walls show a ghost outline of their size. A slab or opening with fewer than 3 points is refused with a message and the points are kept.

New items are placed on the level shown and are numbered and labelled automatically ("Column 3"). Default sizes (SI) are: slab 200 mm, wall 180 mm, column 500 × 500, beam 300 × 600, drop cap 1200 × 1200 × 500. With **Use last modified as default** on (Snap settings dialog, on by default), a new item takes the properties of the last item of the same type whose properties you changed.

Tendons are added with **Tendons & Rebar ▸ Model ▸ Add Tendon** (click the plan points; Enter or a double-click finishes; Esc cancels; then the tendon dialog opens) or with the arrow item *Add Tendon by Values…*. The tendon dialog and tendon editor are described in the tendon part of this manual.

3.2 Supports

The **Support** tool (**Model** ▸ **Add Structural Components** ▸ **Support**; on the Beam tab **Beam** ▸ **Model** ▸ **Support**) places a point support on a beam or a column. It has three types, chosen with the keys **R**, **P** and **F** while the tool runs:

Type	Key	Restrains
Roller (vertical) (default)	R	The vertical movement only
Pin (3 translations)	P	The three translations (it holds the plan position; two or more pins restrain the beam axially)
Fixed (vertical and rotations)	F	The vertical movement and the rotations

Free removes a support. Clicking an existing support opens a dialog with the four choices (Free, Roller, Pin, Fixed). The symbol is drawn at the beam's centre height. A support stored by an older version as "Pin" is now shown as Roller. The Point Supports report has a **Type** column.

3.3 Layers

Layers organise the display and the selection of the model, as in a CAD program. They have no effect on the analysis.

- **Model** ▸ **Selection** ▸ **Layer Settings** (command alias `LA`) opens the dialog **Layer Settings**: one row per layer with the marker of the current layer, **Name**, **On**, **Lock**, **Colour** (AutoCAD index 1 to 255, or "By type"), **Linetype** (Continuous, Dashed, Hidden, Center, Phantom, Dot, DashDot, Divide, Border, By type) and the number of objects. The buttons are **New**, **Delete** (an empty user layer) and **Set Current**. **OK** is one undo step. While the model is locked, only On and Off can change.
- **Model** ▸ **Edit** ▸ **Move to Layer** moves the selected items to a layer. The **Layer** row in the Properties palette and in Item Properties shows the layer of an item, and the **Colour** row after it shows "ByLayer" or an AutoCAD colour (a swatch and its number; "Varies" for a mixed selection). The colour picker has a **ByLayer** choice. A colour change is one undo step and is saved in the layers file.
- **Select by Type** has the criterion **By layer**.
- Layers go into the DXF export (colour, linetype, an off layer as a negative colour, the lock flag) and are stored with the model. They are available for OpenPT (INP-kind) models.

3.4 Snapping

Snaps are set in **Model** ▸ **Snap**, which holds one labelled split button, **Snap** ▾. Its face is the master switch **Snap On/Off** (F3): a click turns snapping on or off, and the face is filled while snapping is on. Its arrow opens a checklist of the snap kinds, with the same keys; the list also holds **Ortho**. **Grid Line Snap** has no button on the ribbon (it is not in the **Snap** ▾ list); it is on by default, works while the master switch **Snap On/Off** is on, and the setting is remembered (`plan.ini` key `snapgridlines`). Each snap is an independent toggle, so several can be on:

Snap (shortcut)	Snaps to
Endpoint (Ctrl+1)	Ends of lines and edges, corners, column/drop centres
Midpoint (Ctrl+2), Centre (Ctrl+3)	Middle of an edge; centre of circle/column/drop, centroid of slab/opening
Intersection (Ctrl+4), Perpendicular (Ctrl+5), Nearest (Ctrl+6)	Crossing of two lines; foot of the perpendicular from the last point; nearest point on a line
Vertices of Components (Ctrl+7)	Stored points: corners, axis ends, centres, tendon points
Grid (F7), Ortho (F8)	Grid: nodes of the snap grid set in Snap settings (only when no object snap is found; this is not the grid lines of the model); Ortho: horizontal/vertical from the last point (Shift = one point)
Grid Line Snap	Intersections of the visible grid lines of the model (see "Grid lines").
Snap On/Off (F3, the face of **Snap** ▾)	Master switch

Defaults: Endpoint, Midpoint, Intersection, Perpendicular, Vertices and Grid Line Snap on; Centre, Nearest, Grid and Ortho off; master on. The settings are remembered. Grid Line Snap is saved in `plan.ini` under its own key (`snapgridlines`), and a `plan.ini` that has no such key starts with it on.

A yellow marker shows the snap point; its shape tells the snap kind (square = endpoint/vertex, triangle = midpoint, circle = centre, X = intersection and grid line intersection, ⊥ = perpendicular, hourglass = nearest, + = grid). A grid line intersection also shows the tip "Grid A/1" beside the marker. When two snaps compete, the priority is Endpoint/Vertex and Grid Line Snap (the closest to the cursor wins), Centre, Midpoint, Intersection, Perpendicular, Nearest.

Snap Settings... (the launcher at the corner of the Snap group) opens **Snap settings**:

Search aperture (pixels, 2 to 20) [5], *Grid origin X / Y* [0], *Grid spacing X / Y* [1000 mm], *Grid angle (degrees)* [0], *Use last modified as default* [on].

Drawn entities (**Model** ▸ **Draw**) of the current level are snap targets too.

3.5 Selecting items

Click selection. In the Plan view (and in the 3D tab) a click selects one item. A click on empty space clears the selection. **Ctrl+click** changes the selection one item at a time:

Ctrl+click on	Effect
An item that is not selected	Adds it. If nothing was selected, it becomes the main (primary) item.
A selected item, the main one or not	Removes it. If it was the main item, the item added last among the others becomes the main item; if none is left, nothing is selected.
Empty space	Clears the selection, as a plain click does.

The grips and the properties window belong to the main item; Modify Selection and the Modify tools act on all selected items. Ctrl+click does not drag a grip and does not edit vertices (the vertex shortcut

is Ctrl+Shift+click, see "Grips and the item context menu"). In the Plan view, a click in Pan mode, with or without Ctrl, selects nothing.

After a Ctrl+click or a selection window the command bar tells what changed and how many items are selected now, for example "Added 1 item, 3 selected." or "Removed 1 item, 11 selected." (also "Removed 1 item, 0 selected." when the last item goes). A plain selection window says "Selected N items."; a Ctrl or Shift window says "Added N items, M selected." (the form is "1 item" for one item).

Model ▸ Selection has these buttons for the mouse modes:

Button	Action
Window Selection/Pick Mode	The default left-button mode: a click picks, a left drag draws a selection window. It is active at start, after New, Open or Import, after a tab or view switch, after Esc and when Pan or Rotate is switched off. Pressing it while it is the home mode does nothing.
Pan	A left drag pans; a click selects nothing. Esc returns to Pick mode.
Rotate	A left drag orbits the 3D view (Ctrl locks one axis). Disabled when no 3D view shows.

Window selection, Pan and Rotate are radio-style toggles: pressing the lit one returns to the default mode. **Esc** in Pick mode clears the selection or cancels a window. Panning also stays on the middle mouse button; in 3D, Shift + middle button orbits. **Redraw** is in **View ▸ Zoom**.

Selection window (in the Plan and in the 3D tab):

- Drag **left to right** (blue, solid outline) = *Window*: selects items lying entirely inside.
- Drag **right to left** (green, dashed outline) = *Crossing*: selects items that touch the rectangle. A crossing window also selects the support lines it touches.
- Hold **Ctrl or Shift** at release to add to the present selection; otherwise the window replaces it (a plain window over empty space clears the selection).
- A window only adds; there is no window that removes items. To remove items, use Ctrl+click or Select by Type with **Remove from current selection**.
- Only visible items of the current level are selected. A very short drag is a click.
- In 3D the rectangle is drawn on the screen; the selection is shared with the Plan, the Properties palette and the Modify commands.

Shared points of a multiple selection. With two or more items selected, a plan position where grips of at least two selected items coincide (within 1 mm) shows a filled square marker; hovering shows "Shared point of <n> items (drag to move them together)". Dragging it moves all those grips together as one edit and one undo step (all or nothing; the first error is shown). It applies to beams, walls, columns, drop caps, slabs, openings, tendons, loads, support lines and splitters, in the Plan and in 3D; not to drawn (CAD) items, section cuts and rebar.

Select by Type

Model ▸ Selection ▸ Select by Type opens the dialog **Select by Type**. It works on the Plan tab and on the 3D tab (see below).

1. In the list, click the item types to select (Beam, Column, Wall, Slab Region, Opening, Drop Cap/Panel, Tendon, Support Line, Splitter, Tributary Region, Point/Line/Patch Load, Rebar, and the drawn types Point, Line, Polyline, Polygon, Arc, Circle, Text, Dimension). Only types present on the current level are listed.

2. Choose a **Criteria**: - **All** (default): every item of the ticked types. - **By keyword**: items whose label (or the level name) contains the text, case-insensitive. `A+B` = must contain both; `A,B` = either. For columns and drops, `RECTANG`, `SQUARE` or `CIRC` select by shape; for support lines, `X DIR` / `Y DIR` select by direction. An empty keyword acts as All. - **By column size**: every column of the level with the chosen size (the ticked types are then ignored, but at least one must be ticked). It is offered only when **Column** is ticked. - **By layer**: every item on the chosen layer (see "Layers").
3. Choose a **Selection Type**: **Refresh all selection** (default), **Add to current selection** or **Remove from current selection**.
4. Click OK. The command bar says "Selected N items." / "Added N items." / "Removed N items." or "No items found."

With nothing ticked the dialog shows "There's no Element Selected to perform the action." Hidden items are not selected. The dialog starts with the defaults every time. Selecting does not change the model and is not an undo step.

In 3D. Select by Type works on the 3D tab with the same window, on the **current level only**; the info line says "Select by Type: N items selected on <level> (current level)". In Multi-Level 3D the other levels stay unselected.

3.6 Grips and the item context menu

When one item is selected and no command runs, small green squares (grips) show at its points: slab and opening corners, beam and wall ends, column and drop centres, tendon control points, plus a centre grip that moves the whole item.

- **Drag a grip** to move that point. Snaps and Ortho apply. Red and green guide lines and dx/dy dimensions show the move; Esc cancels the drag. Connected items do **not** follow.
- **Ctrl+Shift+click on an edge** of the selected slab, opening, support line, splitter or drawn polyline or polygon inserts a vertex there; **Ctrl+Shift+click on a vertex grip** removes it. A slab, opening or polygon keeps at least 3 corners; a support line, splitter or polyline keeps at least 2. This works in the Plan view and in the 3D tab, and acts on the main selected item. A plain Ctrl+click does not change vertices: it adds the item to the selection or removes it.
- A move that makes a beam/wall shorter than 1 mm or a polygon of zero area is refused.

Right-click on an item shows: *Properties...*, *Delete*, *Show in table* (selects the item's tree node; it switches to the Table tab only when that tab is shown), *Fit to item*. For tendons it also shows *Trim/Extend Tendon*, *Display Information...*, **Low Point to Single Parabola** and **Size Tendon...** (see the chapter "Tendons"). **Double-click** opens the properties directly. Right-click on empty space shows *Fit*.

3.7 Item Properties

Model ▸ Modify ▸ Item Properties:

- With a creation command running, it opens the **defaults** of that item type (sizes, offsets, flag above/below, type, angle, material) for the next items.
- With an item selected, it opens the **properties** of that item (as double-click).

The properties window ("<Label> — Properties") shows read-only derived values at the top (length and angle, area and perimeter, etc.) and one editable row per field of the item's INP table. Combos are used

for Flag (0 – below slab / 1 – above slab), column Type (rectangular, circular, generic), the column **Far end** (Default, Fixed or Hinged: the condition of the column end away from the slab; walls are unchanged) and Material. The **Properties palette** (**Model** ▸ **Edit** ▸ **Properties Palette**) shows the same fields in a docked pane; its General rows include **Layer** and **Colour**. For slabs and openings the corners are an editable grid with Add/Remove buttons. **OK**, **Apply** and **Cancel**; invalid values are outlined red and block Apply.

Column and wall end offsets. Columns and walls have a top offset and a bottom offset (`Offset_top`, `Offset_bot`, measured downward from the member's top and bottom plane). OpenPT uses them as in ADAPT to give the member its clear height in the analysis: the top end is at the upper plane minus the top offset, the bottom end at the lower plane minus the bottom offset, with a rigid vertical link to the slab node. Set them in Item Properties or with Modify Selection (Column ▸ Offset, Wall ▸ Offsets).

3.8 Modify Selection (batch edit)

Model ▸ **Modify** ▸ **Modify Selection** opens **Modify Item Properties** for all selected items of the current level (Plan tab).

1. Select the items (click, Ctrl+click to add items or to take one out again, window, or Select by Type).
2. Click **Modify Selection**.
3. Tick the box of each field to change. Fields are greyed until ticked. Each field is pre-filled with the value of the first selected item of that type.
4. Type the new values and click **OK**.

Only ticked fields are written, to every selected item that has that field; other items are skipped. The note "Changes made herein apply to all "selected" components" is shown in the dialog.

Fields: left pane Label (same text to all), Material, FEM and Group (tendons); **Beam** Width/Depth, vertical offset (both ends); **Column** shape (Rectangular/Square/Circular, A, B), angle, top and bottom X/Y/Z offsets; **Drop Cap/Panel** shape (A, B, H), vertical offset; **Wall** thickness, top/bottom offsets; **Slab Region** thickness, vertical offset; **Point Load** Fz, Fx, Fy, Mx, My, offset; **Line Load** loads at both ends, start/end Z (loads only in N-mm files); **Tendon** strand area, duct height, strands, offset, bonded/unbonded, force, end type, seating loss, stressing, estimated long-term stress, friction, minimum radius. Offsets are down positive.

The whole change is one undo step; if one value is rejected nothing is applied. The command bar says "Modified N items." (and how many were skipped).

3.9 Delete, Undo, Redo, Table edits

- **Model** ▸ **Edit** ▸ **Delete** (Del key, or right-click ▸ Delete) removes the selected item.
- **Undo / Redo** (Ctrl+Z / Ctrl+Y) work anywhere except inside a text box being edited. Opening or creating a file clears the undo list. The **Undo** icon is a curved arrow that turns to the left and the **Redo** icon is the same arrow turned to the right.
- **View** ▸ **Table** (for the Table tab; shown only while the Table view tab is active, and hidden by default, together with the Table tab: switch on **Show INP Table/Text tabs (advanced)** to use it, see "Model tree and content tabs"): *Add Row*, *Delete Row*, *Add Element Block* (adds a numbered

block such as SLAB_n, OPENING_n, TENDON_n, AREA_LOADn to the selected container), *Delete Element Block* (later blocks are renumbered). Edits of table cells are undoable too.

3.10 Draw tools (construction geometry)

Model ▸ **Draw** draws plain CAD objects on the current level: **Create Point**, **Create Line**, **Create Polyline**, **Create Polygon**, **Create Circle**, **Create Arc**, **Create Text** and **Create Dimension** (**Ortho**, F8, is not in this group: it is an item of **Snap** ▾; **Create Section Cut** is in **View** ▸ **Section**). They have no structural meaning; use them as construction lines, snap targets and notes. They are saved in the strips sidecar, not in the INP ("Drawings are saved next to the INP, not inside it.").

All draw tools repeat until Esc, `x` or right-click Exit. Points snap and may be typed. Numbers (sides, radius, angle, chord) are typed in the command bar box; where a default is shown, an empty Enter accepts it. The arrow of a split button offers other ways of drawing:

Tool	Default way	Other ways (arrow menu)
Line	first and second point	Continuous Modeling
Polyline	corners; Enter, C, End or double-click finishes (open, at least 2 points)	–
Polygon	corners; finishing closes it (at least 3 points)	Vertices, Centre, Radius · Vertices, Diameter · Vertices, Start Edge, End Edge (number of sides 3–100, default 4)
Circle	Centre, Radius (point or typed value)	Three Points · Diameter
Arc	Start, End, Radius (minor arc, counter-clockwise)	Start, Centre, End · Three Points · Radius, Angle · Radius, Chord
Text	Select Position, Write Text: click point 1, top of font (point 2), direction (point 3), then type the text	Write Text, Select Position
Dimension	start point, end point, position (aligned dimension)	–

Drawn entities can be selected, moved by grips, deleted, copied, mirrored and scaled (see below). Double-click opens "<Kind> Properties" with a **General** tab (coordinates, radius, text, height, angle, justification) and a **Properties** tab (Label, Outline colour `#RRGGBB`, Line thickness 1–8 px). **View** ▸ **Layers** ▾ ▸ **Drawing** shows or hides all drawings. In 3D, drawings are shown flat at the level but cannot be selected.

Drawn entities have layers (see "Layers") but no line styles of their own, fill, groups or blocks. **File** ▸ **Import** ▸ **Import DXF/DWG…** brings CAD geometry in as drawn entities. The DXF export of the structural plan does not contain drawn entities (see "Exporting to Revit, IFC and AutoCAD"). Scale, Move, Copy and Mirror of thousands of drawn items run as one edit with one undo step and do not freeze the window.

3.11 Grid lines

Grid lines are the named reference lines of a plan: for example A, B, C, … in one direction and 1, 2, 3, … in the other. Each line ends in a bubble that holds its name, and the distances between neighbouring lines

are written beside the plan. They help you draw and read the plan. OpenPT does not use them in the analysis, and no item is attached to them.

Do not confuse them with the two other things called grid. The **Grid** row of **View** ▸ **Display** ▸ **Look** ▾ is the adaptive background grid of the Plan view. The **Grid** snap of the **Snap** ▾ list snaps to a regular snap grid (see "Snapping"). Both are only drawing aids.

Grid systems

Grid lines are kept in **grid systems**. This version has Cartesian systems only: straight lines in X and in Y.

- An **X line** stands at one X ordinate and runs in the Y direction. A **Y line** stands at one Y ordinate and runs in the X direction.
- A system has a name, an origin (X, Y), a rotation about Z (counter-clockwise, in degrees), a bubble size and a **Visible** setting. The ordinates of its lines are measured from its origin, in its rotated axes.
- A model may have several systems. The first one is named **GLOBAL**. It has no origin and no rotation (both are 0). Other systems are named G1, G2, ... or as you choose.
- Each line has an **ID**, an **Ordinate**, a **Visible** setting and a **Bubble** setting: **Start**, **End**, **Both** or **None**. **Start** is the end with the lower ordinate of the other direction: the bottom of an X line and the left end of a Y line (in an unrotated system). **End** is the other end: the top of an X line and the right end of a Y line.
- The lines of one direction are kept in order of ordinate. Two lines of one direction cannot stand at the same ordinate.

Grid lines belong to the whole model, not to one level. Every level shows the same grid lines (also in Multi-Level mode), and **Copy Storey** has nothing to copy for them.

Where the commands are

Command	Place
Quick Grids...	Model ▸ Levels & Grids
Grid Systems...	Model ▸ Levels & Grids
Grid Lines (toggle)	View ▸ Layers ▾
Grid Dimensions (toggle)	View ▸ Layers ▾
Grid Line Snap (toggle)	no ribbon button; on by default

The tooltip of each button starts with the full name of the command. **Quick Grids...**, **Grid Systems...**, **Grid Lines** and **Grid Dimensions** need an open file; with no file open they are greyed. **Grid Lines** can also be switched in **View** ▸ **Display** ▸ **View Settings** (row **Gridlines**, column **Display**).

When a model has no grid of its own, OpenPT **derives** grid lines from the columns and walls (see "Drawing standard" in "Start view and what is drawn"). They are not written to any file; **Grid Systems...** turns them into editable lines.

Quick Grids

Quick Grids... makes one system of evenly spaced lines. It opens the dialog **Quick Grids**. Lengths are shown and typed in the length unit of the model, and the unit is in the label of each box, for example

"Spacing X (mm)". Use `.` as the decimal point.

Field	Default	Rule
System name	GLOBAL for the first system; after that G1, G2, ... (the first free name)	Not empty. GLOBAL is only for the first system.
Origin X, Origin Y	0	Any number. For GLOBAL the two boxes are titled **First X ordinate** and **First Y ordinate** and give the ordinate of the first line.
Rotation (deg)	0	−360 to 360. Not used for GLOBAL (the box is greyed).
Number of lines X, Number of lines Y	4	A whole number, 1 to 200.
Spacing X, Spacing Y	24 ft when the length unit of the model is inch or foot; 8 m otherwise (8000 in an N-mm model)	Greater than zero.
X first label	A	See "Label rule".
X labelling	Left to right	**Left to right** or **Right to left**.
Y first label	1	See "Label rule".
Y labelling	Bottom to top	**Bottom to top** or **Top to bottom**.
Bubble size	5 ft when the unit is inch or foot; 1.25 m otherwise	Greater than zero. It is the diameter of the bubble in model length.

If a value is not acceptable, a red text under the boxes says why and the dialog stays open. For example "Number of grid lines must be 1 to 200.", "Spacing must be greater than zero." or "Bubble size must be greater than zero.".

Click **OK**. OpenPT then:

1. Builds the lines. The line i (counting from 0) of a direction is at the first ordinate plus i times the spacing. The first ordinate is 0, or for GLOBAL the value of **First X ordinate** / **First Y ordinate**. With **Right to left** (or **Top to bottom**) the labels run from the highest ordinate down.
2. Puts the bubbles of the X lines at the **End** (top) and those of the Y lines at the **Start** (left). All lines and the system are visible.
3. Switches **Grid Lines** on, and zooms to fit if the new lines and bubbles are not completely in view.
4. Writes "Grid system <name>: 4 x 4 grid lines." in the command bar. The command is one undo step.

If a system with that name exists already, OpenPT asks "Replace grid system <name>?" (**Yes** replaces it, **No** changes nothing). The name GLOBAL is refused with "GLOBAL is reserved for the first grid system." when the model has other systems but no GLOBAL.

Grid Systems

Grid Systems... opens the dialog **Grid Systems**. It edits all systems of the model.

The upper part, **Grid systems**, is the list of systems. Its columns are **Grid system** (the name) and **Visible** (a check box that hides or shows the whole system). The buttons:

Button	Action
Quick Add...	Opens **Quick Grids** (above) and adds the new system. If the name exists, it asks "Replace grid system <name>?".
Copy	Adds a copy of the selected system, named "<name> copy" (then "<name> copy 2", ...).
Delete	Deletes the selected system after "Delete grid system <name>?". GLOBAL can be deleted only when it is the only system; otherwise "GLOBAL can be deleted only when it is the only grid system.".

The lower part, **Grid system data**, shows the selected system:

Control	Meaning
Name	The name of the system.
Origin X, **Origin Y**, **Rotation (deg)**	Position and rotation of the system. They are greyed for GLOBAL.
Bubble size	Diameter of the bubbles, greater than zero.
Display ▸ Ordinates / **Spacing**	Chooses what the second column of the two tables shows (below).
X grid lines, **Y grid lines**	One table for each direction.

Columns of each table:

Column	Meaning
Grid ID	The name of the line. Not empty, at most 12 characters, and used once in the system (X and Y lines together; capital and small letters count as the same).
Ordinate (unit)	The position of the line, measured from the origin of the system. In **Spacing** mode the column is titled "Spacing (unit, row 1: first ordinate)": row 1 shows its ordinate and each other row shows the distance to the row above.
Visible	Whether the line is drawn, counted in the spans and spacing dimensions, and used for snapping.
Bubble	**Start**, **End**, **Both** or **None**.

In **Spacing** mode, a new spacing moves that line and all lines after it by the change, so the other spacings stay. Changing the value of row 1 moves the whole table. Switching to **Spacing** first sorts the tables by ordinate.

The buttons above each table:

Button	Action
Add	Adds a row after the selected row (at the end when none is selected). Its ordinate is the previous ordinate plus the previous spacing (the default spacing of Quick Grids when the row above is the first row; 0 in an empty table). Its ID is the next label after the previous ID, or A (X) / 1 (Y) when there is none or the previous ID cannot count on. Its bubble is **End** for an X line and **Start** for a Y line.
Delete	Deletes the selected rows.
Sort	Sorts the table by ordinate. The IDs stay with their lines.
Relabel...	Opens a small window with **First label** (A for X, 1 for Y) and the check box **Right to left (highest ordinate first)** / **Top to bottom (highest ordinate first)**. It renames all lines of that table in order of ordinate with the label rule.

Click **OK** to accept. OpenPT checks all systems and shows the first problem in red; the dialog stays open until it is cleared. The messages are:

- "Grid ID must not be empty."
- "Grid ID '<id>' is longer than 12 characters."
- "Grid ID '<id>' is used twice in system <name>."
- "Two X lines are at the same ordinate (<value>)." (and the same for Y)
- "Spacing must be greater than zero." (in **Spacing** mode)
- "Bubble size must be greater than zero."
- "Rotation must be between -360 and 360."
- "System name must not be empty.", "System name must be unique." and "GLOBAL is reserved for the first grid system."

On **OK** the lines of every system are sorted by ordinate, and the whole dialog is **one undo step**. The command bar names the systems ("Grid systems: GLOBAL, G1."), or says "No grid systems." when none is left. **Cancel** changes nothing.

Label rule

The first label of **Quick Grids** and of **Relabel...** gives the series of labels. Each next label is made from the one before:

First label	What counts	Series
Letters only (A, ab)	Letters are changed to capitals. The last letter goes up; after Z the letters carry over.	A, B, ..., Z, AA, AB, ...; AZ is followed by BA; ZZ by AAA
Digits only (1, 09)	The number goes up; leading zeros stay.	1, 2, ..., 9, 10; 09, 10, 11; 001, 002
Prefix and number (X1, AX9, Y-01)	The number at the end goes up; the prefix stays.	X1, X2, ...; AX9, AX10; Y-01, Y-02

Anything else is refused with "The first label must end in a letter or a number." Examples that are refused: `1A`, `A1B`, `X-` and an empty label. (`A1` is accepted: it is the prefix A and the number 1.)

The rule applies to the first label of a series. In the tables of **Grid Systems** you can type any ID that meets the rules of the **Grid ID** column.

How grid lines are drawn

The Plan view draws the grid lines over the slabs, beams, columns, walls and tendons, and under the labels, the selection and the snap marker.

- **Lines.** Each visible line is a 1-pixel line. The colour is the colour of the row **Gridlines** in the **Colour Palette** (grey at first). The lines of one direction run from the first to the last visible line of the other direction. If the other direction has fewer than two visible lines, they run across the extent of the model on the plan (and at least to the lines themselves); with no model, 2.5 bubble sizes beyond the lines.
- **Bubbles.** At each end that has a bubble, the line runs on a little further and ends in a circle with the ID inside. The diameter of the circle follows the bubble size of the system, but it is kept between 16 and 64 pixels so that it stays readable. The text of the ID always stays upright, also in a rotated system. Bubbles are staggered: when a bubble would touch the previous bubble of the same side (grid lines 3 and 4, or 10 and 11, close together), it is moved outward by one bubble diameter plus a short leader, and its grid line runs on to it. Zoom Extents leaves room for the staggered bubbles.

- **Spacing dimensions.** For each direction with at least two visible lines, one dimension chain runs along the bubble side: the **End** side if any visible line of that direction has its bubble at **End** or **Both**, otherwise the **Start** side if any has **Start**. A direction with no bubbles has no chain. The chain has a tick at each line and, between neighbouring lines, the distance as text: `8000 (mm)`, `8 (m)`, `24 (ft)`. The number is in the length unit of the model, with 0 decimals for mm, 1 for cm and 2 for other units, and trailing zeros are removed. A text is left out where the gap is too short for it. The text of a vertical chain reads from bottom to top.

Zoom Extents includes the grid lines and their bubbles while **Grid Lines** is on. The 3D view does not show grid lines.

Showing and hiding

Control	Effect	Default
Grid Lines	Shows or hides the lines, bubbles and spacing dimensions. It is the **Display** box of the **Gridlines** row in **View Settings**. **Save as Default** in that dialog remembers it for new models.	On
Grid Dimensions	Shows or hides only the spacing dimensions (chains, ticks and texts). With **Grid Lines** off nothing is shown. It is not remembered between sessions.	On
Visible (per system, in **Grid Systems**)	Hides one whole system.	On
Visible (per line)	Hides one line.	On

Both toggles are greyed with no file open. Quick Grids switches **Grid Lines** on. A `viewitems.json` that was saved before grid lines existed has no choice for them and opens with **Grid Lines** on.

Snapping to grid lines

Grid Line Snap snaps the cursor to the **intersections** of the grid lines: every visible X line with every visible Y line of the same system.

- It is on by default, also for a user whose `plan.ini` is older (the key `snapgridlines` in `plan.ini` holds the choice, 1 or 0).
- It works in the Plan view, with the master switch **Snap On/Off** on, while **Grid Lines** is on. Hidden lines and hidden systems do not snap. The 3D view has no grid line snap.
- It does not snap to points between the intersections, and it does not make the line ends or middles into Endpoint, Midpoint or Perpendicular points.
- It has the same priority as Endpoint and Vertices; the closest point to the cursor wins (see "Snapping"). The marker is the one of Intersection, with the tip "Grid A/1" (X ID / Y ID) for GLOBAL and "G1: A/1" for another system.
- It is a separate snap from **Grid** (F7), which snaps to the nodes of the snap grid.

Storage, undo and lock

- The grid systems of an INP model are stored in the strips sidecar `<model>.inp.strips.json`, under the key `gridSystems`, in millimetres. **Save** writes it (also when the model has no strips), and an .optp project packs it. They are not written into the INP file, the .adm file or the IFC and DXF exports.

- **Quick Grids** and **Grid Systems** each make one undo step (**Ctrl+Z** / **Ctrl+Y**). The display toggles are not undo steps. A grid edit counts as a change of the document (the title shows `*`) until the next Save.
- Grid edits are **allowed while the model is locked**, and so are their Undo and Redo. They keep the analysis results, because grid lines are not analysis input.

Grid lines of a SAFE file

A SAFE file brings its grid lines with it when it is imported (see the chapter "Working with SAFE models"). Whatever the source, the grid lines of an OpenPT model are edited with **Quick Grids...** and **Grid Systems...** as described above, and they are the model's own: nothing is written back to the SAFE file by an export of the model's grid lines.

Limits

Version 1 of grid lines does not have:

- grid systems other than Cartesian (no cylindrical systems, no general lines);
- drawing lines by clicking, adding lines at selected points, dragging a line on the plan, gluing items to the lines, or beams, slabs and support lines drawn along the lines;
- snapping to a point between two intersections, or between lines of different systems;
- grid lines in the 3D view, or in the INP, .adm, IFC and DXF files.

3.12 Modify tools

Model ▸ **Modify** has **Modify Selection**, **Item Properties**, the split button **Copy** ▾ (a click is **Duplicate**; its menu has **Duplicate**, **In-plane**, **Vertical** and **Reference Planes**), **Mirror Selected Items**, **Scale**, **Copy Up** (the command **Columns & Walls Above from Below**), **Delete Point** (the command **Remove Point**) and **Add Point** (the command **Insert Point**). In the group, **Copy Up** stands under **Scale**, and **Add Point** under **Delete Point**. Duplicate, In-plane, Scale, Mirror and Vertical need a non-empty selection (click, Ctrl+click to add or remove items, or window); Insert/Remove Point need exactly one suitable item (one selected item, no others).

Duplicate, In-plane and Mirror act on drawn entities, slab regions, openings, beams, walls, columns, drop panels and loads (Mirror of a load only when its horizontal forces and moments are zero); tendons, strips, section cuts and rebar are skipped and counted ("Copied 5 items; skipped 2 (not supported: tendon)"). Scale acts only on drawn entities (circles/arcs only with equal factors; text and dimensions are not scaled).

Duplicate

1. Select the items, click **Model** ▸ **Modify** ▸ **Copy** ▾ ▸ **Duplicate**.
2. "Enter the translation base reference point": click or type the base point.
3. "Enter the insertion point": each click makes one more copy of the whole selection, moved by (insertion − base). The prompt repeats.
4. Esc ends the command and clears the selection. Each copy is one undo step.

In-plane (copy or move)

1. Select, click **Copy** ▾ ▸ **In-plane**.

2. "Start point of the vector", then "End point of the vector".
3. The dialog **Copy - Move** shows **Offset X**, **Offset Y** (from the vector, editable) and **Number** (copies, 1–100, default 1).
4. Click **Copy** (Number copies at 1×, 2×, ... the offset) or **Move** (moves once; Number is ignored), or **Cancel**. One undo step.

Moving structural items uses the grip-move rules; if one item cannot be moved, the others still move and the message names the refusal.

Scale

Select drawn polygons, polylines, lines (structural items cannot be scaled), click **Model ▸ Modify ▸ Scale**, enter **In X direction** and **In Y direction** (default 1, must be > 0) in **Scale Factor**, OK. Each item is scaled about its own last vertex (circles and arcs about their centre). If only structural items are selected, a message explains that the tool does not apply to structural components.

Mirror Selected Items

Click **Mirror Selected Items**, then "Start point of the axis" and "End point of the axis". A mirrored copy is created; the original is kept. Column and drop angles are mirrored; text stays readable. One undo step.

Insert Point / Remove Point

For exactly one selected drawn polyline or polygon, slab region, opening, support line, splitter or tributary:

- **Insert Point** (the button **Add Point**): "Select vertex to insert vertex". Clicking a vertex inserts a new vertex at the midpoint of the segment that ends at that vertex; clicking on an edge away from vertices inserts at that point.
- **Remove Point** (the button **Delete Point**): "Select vertex to delete". A polygon, slab region, opening or tributary keeps at least 3 points, a polyline, support line or splitter at least 2.

Both repeat until Esc. Without either command you can do the same with **Ctrl+Shift+click** on an edge (insert) or on a vertex grip (remove) of the selected item (not for a tributary; see "Grips and the item context menu"); the tooltips of the two buttons say so.

Not available

ADAPT's By Coordinates, Rotation, Trim, Extend, Join, Split, Blocks and clipboard Cut/Copy/Paste are not part of OpenPT.

3.13 Columns and walls above the slab from the ones below

Model ▸ Modify ▸ Columns & Walls Above from Below copies every column and wall below the slab to the side above the slab: same position, section, size, rotation, material and far-end fixity; the height is that of the level above, else that of the member below. It acts on the shown level, which must have no level above (the top level); with a selection, only the selected members are copied; members that already exist above are skipped. It is one undo step; the info line says "Created N columns and M walls above the slab (K already there)." It is refused while the model is locked.

3.14 Vertical copy / move (to other planes)

Model ▸ Modify ▸ Copy ▾ ▸ Vertical copies or moves the selected items to other planes, for example to add columns and walls above the slab.

1. Select the items on the current level.
2. Click **Copy ▾ ▸ Vertical**. The dialog **Copy/Move Vertical** reads as one sentence: `[Mode] the selected items [Direction] [Plane | Count] time(s) to existing planes.`

Mode: Copy [default], Move, Assign. Direction: To, Up [default], Down (Move and Assign always use To). Plane (for To; top plane by default) or Count (for Up/Down, ≥ 1, default 1). 3. Click OK.

Copy To P makes one copy on plane P; **Copy Up/Down n** makes n copies on the next n planes. **Move To P** moves the items to P with unchanged offsets (they move with the plane). **Assign To P** moves them to P but keeps their absolute elevation by changing their vertical offsets; items that would get zero or negative height are refused.

Notes:

- A column or wall stored on plane t stands between plane t−1 and t, so a copied column or wall is one storey tall and takes that storey's height. A column or wall cannot go to the bottom plane.
- Items whose target plane does not exist are dropped (no planes are added); the message says how many.
- Tendons, support lines, splitters and rebar are copied too (labels as in Copy Storey).
- If a copied column or drop cap lands on an existing one, OpenPT asks once: "A Column already exists at this location. OK to continue paste. Cancel to cancel paste." Cancel aborts the whole command.
- The selection stays; copies are not selected. One undo step per command.

3.15 Trim/Extend Tendon

Tendons & Rebar ▸ Modify ▸ Trim/Extend Tendon (also right-click on a tendon; on the Beam tab **Beam ▸ Tendon ▸ Trim/Extend Tendon**) moves the ends of the **selected tendons** onto the nearest slab or opening edge, when that edge is within the trim tolerance.

1. Optional: **Tendons & Rebar ▸ Settings ▸ Tendon Settings...** sets **Trim Tolerance** [m] (default 0.500 m; accepted 0.001 to 100). It is remembered for your user account.
2. Select one or more tendons.
3. Click **Trim/Extend Tendon**.

Rules:

- Each end slides along the line of its own end span (not perpendicular to the edge). An end that stops short of the edge is extended; an end that sticks out past it is trimmed back.
- The distance is measured along that line. An end 400 mm from an edge at 45° (566 mm along the line) does not move with a 500 mm tolerance.
- Edges are the edges of slab regions and openings of the level; beams, walls and columns are not edges. The nearest edge wins.
- Only the plan position of the two end points changes. Heights, profiles, strands, forces and interior points are kept. An end never moves past the other end of its span.

- The command bar says "Trimmed/extended <n> ends of <m> tendons." or "No tendon end is within <tol> mm of an edge." One undo step; pressing it again changes nothing.

3.16 Parallel and radial support lines

Design ▸ Strip Modelling ▸ Parallel Support Lines draws a family of parallel support lines across the slab.

1. Click **Parallel Support Lines**.
2. Enter **Spacing** [mm] in the dialog (default: the last value, first time 1000; accepted 1 to 1 000 000). OK.
3. Draw the first support line: click its vertices (2 or more); Enter, C, End or double-click finishes it.
4. "Click the side where the parallel lines go." A dashed preview shows the lines on the cursor's side. Click to create them.

The lines are the first line offset by 1×, 2×, ... the spacing, clipped to the slab regions (a line leaving the slab is split; lines are kept across openings). It stops when an offset line no longer touches the slab. Above 200 lines OpenPT asks "<n> support lines will be created. Continue?". All lines take the direction of the first line (X when its |dx| ≥ |dy|, else Y) and the support-line defaults. One undo step. Esc at any step creates nothing.

Radial Support Lines (a command without a ribbon button in this build; it keeps working for fan-shaped slabs when it is called by a keyboard alias you assign in Keyboard Shortcuts) works the same way:

1. Enter **Angle step** [deg] (default: last value, first time 5; 0.01 to 90).
2. "Radial support lines: click the centre point (the apex)." (It may lie outside the slab.)
3. "Click a point on the first line (its direction from the centre)." The first line runs from the centre through the slab.
4. Click the side (clockwise or counter-clockwise). Lines are created every step, clipped to the slab, until a ray no longer meets the slab or 360° is reached. Each line's direction (X/Y) is its own dominant axis.

3.17 Beam support lines

Beam ▸ Design ▸ Beam Support Lines puts a support line along each beam of the level and then generates the design sections, so the beams are designed as strips.

1. Click **Beam Support Lines**. Beams that run in a straight line are one line through their columns; beams that meet at an angle are separate lines. Beams shorter than 300 mm are left out. On a SAFE F2K, wall lines and lines shallower than the slab are not beams.
2. A beam that already has a support line along it gets no new one. A line counts as along the beam when it is nearly parallel (within 5°), within half the beam width (at least 150 mm) of the beam axis, and covers at least half of the beam's length. Your own lines are kept.
3. Each new line takes the criteria **Beam** and the options of **Support Line Defaults...**. It runs along the beam; an end inside a column goes to the column centre. The strip is the tributary, half way to the next support line, so the beam is designed with its slab and the tendons in the strip count.
4. OpenPT then generates the tributaries and design sections of the whole level again. If the level has manual tributary regions, it asks first, because Generate Sections replaces them.

The info line says how many lines were added and skipped, and how many tributaries and design sections were made. One undo step. The command is refused in Multi-Level mode. The command does not run the analysis: run it, then **Design the Sections**. For criteria **Beam** the support line dialog has a **Design Width Limit** "Effective flange" (see the chapter "Design").

The other strip modelling tools are described in the chapter "Design".

3.18 Column positions near drop panel edges (meshing note)

When **Mesh Generation** (**Analyse** ▸ **Meshing** ▸ **Mesh Generation**) shifts nodes automatically (option *Shift nodes automatically*, with *Maximum shift distance*), a column that has no beam or wall ending on it and lies within the shift distance of a drop panel edge is moved onto that edge (the foot of the perpendicular), as ADAPT does. Drop panel edges are not targets for beam and wall ends. You do not need to do anything; this note explains why a column node in the mesh can sit up to the shift distance away from the drawn column centre.

4 Multi-level models

A model's **planes** (reference planes, the INP's LEVEL_1, LEVEL_2, ...) are listed in the level combo as "Level 2 (z = 3.30 m)" or by the level's own name. The first plane is the bottom. A column or wall stored on plane k stands between plane k−1 and plane k. A normal ADAPT single-floor model has three planes: bottom, current (with the slab) and top.

4.1 Single-Level and Multi-Level mode

The two buttons to the right of the level combo switch the mode:

- **Single-Level** (default): "view, model and analyse the current plane only".
- **Multi-Level**: "view the entire structure; analyses consider all levels". The 3D view shows every level, the plan shows the current plane with the others greyed, and editing still acts on the current plane only.

Switching the mode clears the selection. Every newly opened document starts in Single-Level mode.

In Multi-Level mode, **Copy Storey**, **Generate/Design Sections**, the shear (punching) check, **Cracked Deflection**, **Generate Rebar** and **Beam Support Lines** are refused with "Functionality not allowed in multi-storey view."

4.2 Story Manager

Model ▸ **Levels & Grids** ▸ **Story Manager** opens the dialog **Story Manager**.

The list shows **Name | Elevation | Height** (top plane first; Height = distance to the plane below, 0 for the bottom plane). Under the list: **Label** and **Height** boxes for the selected row, and the buttons **Apply**, **Add**, **Delete**, **Set as active**. The active plane is named under the list.

Action	What it does
Apply	Renames the plane (Label) and/or sets its storey height. The plane and all planes above move by the difference, so the other storey heights are kept. Loads on moved planes move with them. Height must be > 0.
Add	Inserts an empty plane right above the selected one at its elevation + Height (default: the selected plane's storey height, or 3000 mm above the bottom plane). Higher planes rise by that height. The new plane is named "Plane N". Columns of the split storey keep their planes.
Delete	Asks "Do you want to delete permanently all the Structural Components of this reference plane?" Yes deletes every item of that plane (and its strips and drawings) and removes the plane; the planes above drop by its storey height. The bottom plane cannot be deleted, and a model keeps at least 2 planes.
Set as active	Makes the selected plane the current one.

OK keeps all changes as one undo step; **Cancel** undoes them all.

4.3 Copy Storey

Model ▸ Modify ▸ Copy ▾ ▸ Reference Planes ("Copy Storey") copies the current plane with all its components to new planes, to build typical floors quickly. Single-Level mode only.

1. Show the plane to copy (level combo).
2. Click **Copy ▾ ▸ Reference Planes**.
3. In the dialog **Copy Storey** enter:

Number of copies [1], **Distance (storey height of each copy)** [the current plane's storey height], **Direction** Up [default] or Down.

4. Click OK. One undo step.

Up: for each copy a new plane is inserted right above the last one at +Distance (higher planes rise by Distance), and every item is copied onto it: columns and walls (one storey, ending at the new plane), beams, slab regions, drop caps/panels, openings, loads (each in its own load case), supports, springs, tendons (profiles relative to the slab), base and mesh rebar, support lines and splitters.

Down: each new plane is inserted right below the source; the source and the planes above rise by Distance; the copy goes onto the new plane. Not possible from the bottom plane ("There is no storey below the bottom plane: Copy Storey Down is not possible from it.").

Not copied: drawn entities, tributaries and design sections (generate them per level), column end releases, slab meshes. Item labels are copied unchanged ("Column 12" on every floor). New planes are named "Plane N". Grid lines are not copied because they belong to the whole model; every plane shows them (see "Grid lines").

Copying with **Vertical** (Copy Up 1 of everything) gives the same result on existing planes.

4.4 Analysis of a multi-level model

- **Execute Analysis** (F9) in Multi-Level mode analyses all slab levels in **one** FE model: each slab is meshed as in a single-level run, and the columns and walls of each storey connect the slab below to the slab above. Only the base (members whose lower plane has no slab) is restrained, by default fixed in all six directions; the support set is an option in the Criteria window (Analysis/Design Options). The same support check as for one level applies to the assembled model (see "Support check" in the chapter "Meshing and analysis"): a part held only in its plane gets stabilisers and a warning, a part free out of plane stops the run with an error, and a level with no supported part is refused with a message.
- Load cases of all levels form one list (by name); self-weight covers all slabs, columns and walls; all tendons give one Prestressing case.
- Results: contours and diagrams show the current plane's part of the global solution; Level Up/Down switches the level without a new solve; the 3D view shows the deformed shape of all levels. In the Results View tree global results are marked "(G)"; a **Member forces (G)** tab shows the member forces of the whole model.
- Strip design, punching, cracked deflection and the calculated rebar stay single-level: in Single-Level mode they use the single-level analysis of the current plane.
- Results of a multi-level run are saved in the results store like single-level results.

4.5 Saving and exporting multi-level models

- **.optp:** a project holds any number of planes.
- **INP** (File ▸ Export ▸ ADAPT model (.INP)...): any number of planes is written in ADAPT's own layout. ADAPT-Builder imports such a file ("Create new model", all levels).
- **.adm** (File ▸ Export ▸ ADAPT-Builder model (.adm)...): the .adm writer holds exactly three planes. For a model with more planes OpenPT says "This model has N planes. The ADAPT project (.adm) writer holds exactly 3 planes. Use Export and choose the INP format instead".

5 Command-line use

`OpenPT.Gui.exe` has switches for batch work: pictures of models and results, conversions, exports, analyses and projects, without the window. The switch must be the **first** argument; otherwise the first argument is a file to open in the window (an `.optp` is opened, any other model file is imported).

- Messages go to the console the program was started from. Exit code 0 = success, 1 = operation failed, 2 = wrong arguments or error (the usage list is printed; an unknown switch prints "Unknown switch ..."), 3 = no valid licence (a ship build).
- `<LEVEL_n|auto>` is a level name such as `LEVEL_2`, or `auto` for the first slab level. Pictures are PNG; `[width height]` in pixels is optional. Quote paths with spaces.
- `--lang en|vi` is the one option that may stand anywhere on the command line, also before the switch or in front of a file to open. It sets the interface language for that run (see "Interface language"); it is not saved. A run that starts with a switch uses English unless `--lang vi` is given, whatever language is saved in `plan.ini`.
- `--progress` on `--meshgen`, `--render-design`, `--render-punching` or `--cracked` prints one line per 5 % of the run to the error output, for example `progress 45 % Factorising the stiffness matrix`.
- Which switches a given build lists in its usage text depends on the build; a switch named here that your build does not know prints "Unknown switch".

```
OpenPT.Gui.exe --render-plan "D:\Jobs\B01\B01.INP" auto D:\out\plan.png 2400 1600 --labels
OpenPT.Gui.exe --pack-optp "D:\Jobs\B01\B01.INP" D:\Archive\B01.optp
```

5.1 Files and projects

Switch	Arguments	What it does
`--pack-optp`	`<model file> <out.optp> [--no-results]`	Packs a model, its sidecars and (unless `--no-results`) its results into an .optp.
`--unpack-optp`	`<in.optp> <out folder>`	Unpacks an .optp into loose files.
`--roundtrip`	`<in.inp> <out.inp>`	Reads and rewrites an INP (read/write check).
`--f2k-to-inp`	`<in.f2k> <out.INP> [options]`	Converts a SAFE text file into an INP plus sidecars and prints a completeness report.
`--export-f2k`	`<in> <out.F2K> [--level <name>] [--safe12]`	Writes a SAFE text file (SAFE 20 and later table format; `--safe12` writes SAFE 12).
`--export-inp`, `--export-adm`, `--export-fdb`	`<in> <out>`	Write an ADAPT exchange file, an ADAPT-Builder project or a SAFE 22 database.
`--elongation`	`<in.INP> <out folder> [--params p.json] [--method table|profile] [--compare old.xlsx]`	Tendon elongation table (xlsx, PDF, `_checks.txt`); see the chapter "Tendons".
`--export-ifc`	`<model.inp> <out.ifc>`	Writes every level of a model as an IFC file.

Switch	Arguments	What it does
`--export-dxf`	`<model.inp> <out.dxf> [--level all|<n>[,<n>...]] [--per-level-files]`	Writes the plan of one level, of the listed levels or of all levels as a DXF file; `--per-level-files` writes one file per level.
`--export-revit`	`<model> <out.optrvt.json>`	Writes the exchange file for the OpenPT add-in of Revit.

`--f2k-to-inp` options (defaults in brackets): `--height-below` / `--height-above <mm>` storey heights [3300]; `--adapt-columns` columns below the slab instead of SAFE-like point supports, `--columns-above` also above; `--walls` wall lines become INP walls, with `--wall-min-depth <mm>` [3000]; `--column-size <mm>` for columns missing from the columns sidecar [400]; `--no-openings`; `--openings FILE`, `--columns FILE` [the `<stem>.openings.json` / `.columns.json` next to the SAFE file]; `--code` [EC2; TCVN is accepted]; `--cell <mm>` [600]; diagnostics `--solve CASE` and `--compare [--only A,B | --no-patterns]`.

`--export-ifc`, `--export-dxf` and `--export-revit` write the same files as the File ▸ Export commands (see "Exporting to Revit, IFC and AutoCAD"). They open no window and print the summary line of the export to the console. `<n>` is the number n of the level name `LEVEL_n`; without `--level`, the DXF takes the level with the most elements (the level that the Plan opens on). The exit code is 0 on success; 1 when the input is not found or cannot be read ("Input not found: <path>", "Cannot read <path>: <reason>"), when the level does not exist ("Level not found: <n>") or when the file cannot be written ("Cannot write <path>: <reason>"); and 2 for wrong arguments, when the usage line is printed.

5.2 Analysis and design

Switch	Arguments	What it does
`--modal`	`<model> <out.csv> [--modes N] [--mass "SW=1,SDL=1,LL=0.1"] [--lateral] [--cracked <combo>] [--shapes <dir>]`	Modal analysis (default 3 modes); writes the modal table as CSV and, with `--shapes`, the mode shapes. See "Modal analysis".
`--beam-design`	`<model.inp|.f2k|.optp> <out folder> [--code TCVN|EC2|ACI14] [--stirrup 10] [--legs 2]`	Beam design; writes `beam_design.csv`, `beam_sections.csv` and `beam_design.log`.
`--section-check`	`<cases.json> [out.csv]`	The Section Check engine on a list of cases (`"code"` may be `ACI14` or `EC2`); exit code 0 if no row is DIFF or ERROR. A developer harness.
`--cracked`	`<file.inp> <level|auto> <cell mm> <out.png> [w h] [--adm <file.adm>]`	Cracked long-term deflection; it still needs strips.

5.3 Pictures

Switch	Arguments	Output
`--render-plan`	`<in.inp> <LEVEL_n|auto> <out.png> [w h] [--labels] [--tab <name>]`	Plan of one level, drawn as the Model tab (no loads) unless a tab such as `--tab Loads` is given

Switch	Arguments	Output
`--render-3d`	`<in.inp> <LEVEL_n|auto> <view> <out.png> [zscale] [w h]`	3D view; `view` = iso, top, front, right, left, rear, bottom or `a=<deg>,e=<deg>`; the picture is shaded unless `--wire` is given
`--render-tendon`, `--render-tendon-dialog`	`<in.inp> <TENDON_k|index> <out.png | out prefix>`	Tendon profile / tendon dialog
`--render-window`	`<in.inp> <out.png> [plan|table|text] [w h] [node path] [--common-tab T|all]`	The main window; node path e.g. `DATA_OF_EACH_LEVEL/LEVEL_2/GEOMETRY/COLUMN`
`--render-cut`	`<in.inp> <out folder> --level <n> --cut x1,y1,x2,y2 --at x,y [--label "text"]`	Section cut drawing
`--render-criteria`	`<in.inp> <out folder>`	Criteria dialogs
`--render-strips`	`<in.inp> <in.adm> <out prefix>`	Strips imported from an .adm
`--meshgen`	`<file.inp> <level|auto> <cell mm> <out.png> [w h] [--case pt|service|hyp|strength|all]`	Mesh and deflection of one case (default: all loads + self-weight, no prestress)
`--render-results`	`<in.inp> <in.adm|-> <out folder> [--dsi <file.DSI>]`	Deflections, strip and beam diagrams
`--render-design`, `--render-punching`	`<in.inp> <level|auto> <cell mm> <out folder>` (design also `[--adm f]`)	Section design / punching check
`--render-rebar`	`<in.inp> <LEVEL_n|auto> <out folder>`	The rebar of the model (it draws the bars that exist)
`--render-f2k`, `--render-f2k-design`, `--render-f2k-3d`	`<in.f2k> <out folder> [...]`	A SAFE model and its results, design and 3D pictures
`--render-icons`	`<out folder> [family]`	Review sheets of the command icons
`--dump-menus`	`<out.md>`	The command table: every command with its place on the ribbon

`--common-tab <name>|all` shows that ribbon tab (`all` = one picture per tab); the names are the tab names of this manual, for example `Analyse`.

`--render-plan` and `--render-rebar` also draw the grid lines stored in the strips sidecar of the INP (see "Grid lines"), with **Grid Lines** and **Grid Dimensions** on, and fit the picture so that the bubbles are inside it.

5.4 Developer and diagnostic switches

`--analyze <deck.inp> <lin|sw|crk> <out.png>` solves an ADAPT analyser deck (lin = no self-weight, sw = self-weight, crk = with cracks); `--admlive <in.inp> <out.adm> [design|summary|vertical|...]` meshes, solves and saves an .adm as Export does, answering Yes to every question; `--render-ribbon <out folder>`, `--render-theme <out.png>`; `--snap-bench <in.inp> <level> [queries]` and `--bench-nav` time snapping, panning and zooming; `--bench-analyze` times the analysis of a model. `--read-safe22`, `--read-fdb`, `--write-fdb` and `--edit-safe22` are developer gates of the SAFE readers and writers. `--lang-collect <file>` (with Vietnamese, `--lang vi`; it may stand anywhere on the command line like `--lang`) appends, once each, the texts that the interface drew without an entry in `lang\vi.txt`, as lines `English text =` for you to fill in.

5.5 Self-tests

The program contains built-in self-test switches (`--selftest...`) that run checks headless and print PASS/FAIL lines. They are meant for the people who develop and install OpenPT, they can take several minutes, and they are not needed for daily work. They write only into the work folder you give them.

6 Tendons

This chapter covers everything OpenPT does with post-tensioning tendons: how a tendon is defined and edited, how its profile is built, how friction, anchor seating and long-term losses are calculated, how the tendon is turned into loads on the slab, and the tools that help to lay a tendon out, and the tools that report tendon data for site use (display, chair heights, elongation table).

6.1 Where the tendon tools are

Place	What it holds
Tendons & Rebar ▸ Settings	**Tendon Settings...** (trim tolerance), **Chair Settings...**
Tendons & Rebar ▸ Model	**Add Tendon** (click points); the drop-down gives **Add Tendon by Values...**
Tendons & Rebar ▸ Modify	**Tendon Editor**, **Trim/Extend Tendon**
Tendons & Rebar ▸ Shop Drawing	**Elongation Table...**
Visibility group of the Tendons & Rebar tab	**Show Tendons**, **Tendon Display Manager**
Design ▸ Criteria	**Materials** (the prestressing steel materials among them) and **Allowable Stresses**
View ▸ Layers	**Tendons** (a toggle), and in the **Layers** ▾ list **Tendons** and **Tendon Heights**; **Group Colours** is in **View ▸ Display ▸ Look** ▾
Beam ▸ Tendon	**Add Tendon**, **Trim/Extend Tendon** (the Beam tab; see "Beams without a slab")
Right-click menu of a tendon (Plan and 3D)	**Properties...**, **Trim/Extend Tendon**, **Display Information...**, **Low Point to Single Parabola**, **Size Tendon...**

The tendon list, the elevation profile and the force diagram that earlier versions showed in a Tendons document tab are in the **Tendon Editor** (its Shape/System/Friction tab) and in the Properties palette. Tendons are named "Tendon 7" in the window; the file keeps the identifier `TENDON_7`.

6.2 Adding a tendon

By clicking (**Tendons & Rebar ▸ Model ▸ Add Tendon**):

1. Click the plan points of the tendon in order. Each click ends one span.
2. Press Enter or double-click to finish. Esc cancels.
3. Snaps and typed coordinates (`x,y`, `@dx,dy`, `@length<angle`) work as in the other drawing commands.

By values (the arrow under Add Tendon ▸ **Add Tendon by Values...**): a dialog "Add tendon to <level>" with a point grid, a span grid and these general fields:

Field	Default
Label (blank = automatic)	blank
Group	1

Field	Default
Strands	4
Strand area (mm2)	98.7
Duct height (mm)	6
Stress method 3: jacking / fpu	0.8
Live end	Both ends (choices: First end, Last end, Both ends)
Seating (mm)	6
Long-term loss (MPa)	150
Friction mu	0.06
Wobble K (1/mm)	0.000004

When a tendon is selected before the command, its values are used as the template instead of these defaults.

6.3 The tendon window (Tendon Editor)

Open it with **Tendons & Rebar ▸ Modify ▸ Tendon Editor** (with a tendon selected in the Plan), or by double-clicking a tendon in the Plan or 3D view. The window is titled `<label> — Tendon`. It is modal.

Like every dialog, it ends with **Apply**, **OK** and **Cancel** at the bottom right. **Apply** writes all changes as one undo step and redraws; **OK** applies and closes; **Cancel** discards changes that were not applied (it asks first if there are any).

Tabs: **General**, **Stressing**, **Location**, **Shape/System/Friction** (opened first) and **Fields** (the raw name/value list of the tendon's general data).

General tab

Label, Group, Material (the prestressing materials of the model), Area per strand (mm^2), Number of strands, Duct height, System name, Mode (Straight/Spline/Fillet as stored), and the read-only BuilderID and GUID.

Stressing tab

- **Method:** Calculated force or Effective force.
- **Effective force per strand** (kN), used only by effective-force tendons.
- For the **First End** and the **Last End**: Live or Dead, Seating loss (mm), stress method (Force given / Stress given / Stress/fpu), the value, and the anchor fields as stored.
- **Long-term loss**: **Lump sum** (default; the loss in MPa applied after seating) or **Calculated** (see "Long-term losses" below).

Location tab

One row per span joint with X (mm) and Y (mm). The column "Z (mm), slab top" is read only. Editing a point moves that joint exactly as dragging its grip in the plan does.

Shape/System/Friction tab

From top to bottom:

1. **Profile chart** with the radio buttons **Shape Diagram** and **Force Diagram**.
2. **Span grid.** Columns: Span, Shape, a small shape icon, Length (m, read only), Depth at first support, Height at low point 1, Height at low point 2, Depth at last support, Inflection 1 (fraction of span), Low point (fraction of span), Inflection 2 (fraction of span), Harp flat (fraction of span), Friction coefficient, Wobble coefficient (rad/m) and System (Bonded/Unbonded). The depth and height headers carry the length unit; a tooltip on each says what is measured. The first row, **Typical**, writes its value to every span where that cell applies (it starts from the most common value of each column and is never saved).
3. Below the grid: **First Span / Last Span** with **Insert** and **Delete**, and the box **Minimum radius of curvature (R):** [m].

Units in the grid: Length and R in m (3 decimals), the depths and heights (the CGS) in mm, the inflection, low-point and harp-flat values as plain fractions of the span length, the friction coefficient as a number, wobble in rad/m (the INP stores it per mm, so the grid shows 1000 × the file value), uplift in kN/m.

CGS convention. The **Depth** values (at the first and the last support) are measured **down from the slab top**; the **Height** values (at the low points) are measured **up from the soffit**. The soffit is the slab top minus the thickness of the slab region under the point.

Shape list (in this order): Reversed Parabola, Partial Parabola, Harped, Straight, Extended Reversed Parabola, Cantilever Up, Cantilever Down, Cantilever Partial, Cantilever Harped. The cantilever shapes are offered only for the first and the last span. Only the cells that a shape uses are enabled:

Cell	Used by
Inflection 1 (X1)	Reversed, Partial, Harped, Cantilever Partial, Cantilever Harped, Extended Reversed
Low point (X2) and Height at low point 1	Reversed, Partial, Harped, Extended Reversed
Height at low point 2	Extended Reversed
Inflection 2 (X3)	Reversed, Partial, Harped
Harp flat (A)	Harped
Depth at first support, Depth at last support	all shapes
Friction coefficient, Wobble coefficient	read only for effective-force tendons

When you change a span's shape, the fractions are reset to Inflection 1 = 0.1, Low point = 0.5 (0.1 for Extended Reversed), Inflection 2 = 0.1, Harp flat = 0.1; the covers are kept. Editing a span's Depth at first support also sets the previous span's Depth at last support, and the other way round, so the profile stays continuous over supports.

Validation. Invalid fractions turn the cell red, and Apply is refused with "The ratios for the red cells do not satisfy the tendon profile requirement." The rules:

Shapes	Valid when
Reversed, Partial	$X1 + X3 \le 1$ and $X1 \le X2 \le 1 - X3$

Shapes	Valid when
Harped	X1 + X3 + A ≤ 1 and X1 + A/2 ≤ X2 ≤ 1 − A/2 − X3
Extended Reversed	X1 + X2 ≤ 1
Straight and cantilevers	always valid

Insert / Delete. Insert splits the chosen end span at its plan midpoint; the outer half becomes the new end span. Delete removes the chosen end span and moves its outer cover to the new end span. Both are refused on a span with swerve points, while there are invalid (red) ratios, and Delete is refused when only one span is left.

Shape Diagram. Draws the slab top and soffit along the tendon (green), span boundaries (grey dashed), the tendon (dark red) from the unapplied values, a blue tick with the value at each CGS control point, the radius labels `R=…` (red when below the minimum radius box), "Span n / L=x.xx" under each span, and an **Uplift (kN/m)** row below the chart. The live and dead end markers are drawn as in the plan. The mouse wheel zooms horizontally, a middle-button drag pans and a double-click fits.

Force Diagram. fps/fpu along the tendon from the friction and seating calculation, with "Maximum = …, Average = …", and "Jacking Ratio = …" and "Elongation at the first end = … mm" at the first live end (and "Elongation at the second end (added) = … mm" at a second end that is jacked after the first). It is disabled for effective-force tendons.

Radius of curvature. For each parabolic piece of length a and rise h: `R = a² / (2·|h|)`. At a low point the smaller R of its two pieces is shown; at an interior support the smaller of the two spans' end pieces. The **Minimum radius of curvature** default is 2.0 m; it is kept per user, not in the file, and only changes the colour of the labels.

Uplift. The average upward deviation force of a span: `Uplift = P · Σ (2·d_i / ℓ_i) / L_xy`, summed over the upward-curving parabola pieces (plan length ℓ_i, rise d_i), with L_xy the span's plan length. P is the effective force per strand × strands. For calculated-force tendons, whose file has no effective force, the dialog uses the average force from the friction calculation.

6.4 Profile geometry

OpenPT builds the 3D strand path (CGS) of every tendon span by span from the plan points, the shape, the covers and the fractions. Every span is straight in plan between its two plan points.

Control points (Reversed Parabola). High point A at the span start (z = slab top − Top First), low point L at X2·L (z = soffit + Bottom 1), high point B at the span end (z = slab top − Top Last). The inflection points lie at X1·L from the start and X3·L from the end, on the straight lines A–L and L–B. The fractions are fractions of the **whole span**. Four parabolic pieces are used, each with a horizontal tangent at its high or low point, so the slope is continuous at the inflection points.

Other shapes:

Shape	Geometry
Partial Parabola	flat at the end heights up to X1·L and from L − X3·L, parabolas to the low point at X2·L (slope kink at the change)
Harped	straight legs; a flat bottom of length A·L centred at X2·L

Shape	Geometry
Straight	one straight line between the two end heights
Extended Reversed Parabola	reversed parabolas in the two end zones (lengths X1·L and X2·L), straight between the two low points
Cantilever Up	one parabola over the span with its vertex at the free end
Cantilever Down	one parabola with its vertex at the support
Cantilever Partial	straight from the support for X1·L, then a parabola with its vertex at the free end
Cantilever Harped	straight from the support for X1·L, then straight to the free end

Segments. Each piece is divided into straight segments for the friction calculation: a piece of 3D length ℓ in a span of length L gets `n = ⌈10·ℓ/L⌉ + 1` segments (at least 1). Straight legs of the harped and partial shapes get one segment each. Each segment carries its span's μ and K.

A tendon over a beam. Where a tendon span crosses a beam at its low point, the low point does not drop towards the beam soffit (it is measured from the beam soffit only where the tendon lies in the beam's plan width), as ADAPT reports the tendon lengths; the analysis and the tendon reports use the same rule. Tendon lengths and results of models with such spans differ slightly from earlier versions.

Bonded and unbonded. The system flag of a tendon is read as ADAPT reports it: 0 = bonded, 1 = unbonded.

Limitation. Tendons curved in plan (swerve points, spline or fillet modes) are drawn, but their friction geometry uses the straight plan chord of each span. The slope of the slab top along a span is not modelled; a span keeps a flat tendon plane between its end tops.

6.5 Friction, anchor seating and elongation

These results appear in the Force Diagram of the tendon window, in the texts of the Tendon Display Manager and in the Elongation Table. In the Tendon Editor the radio buttons follow the diagram shown; the elongation labels read "Elongation at the first end = N mm" and "Elongation at the second end (added) = N mm"; numbers read `R = 98.0 m` and `Span 1 / L = 8.40 m`.

Jacking force. P0 = the given force, or stress × A, or (stress/fpu) × fpu × A, per live end.

Friction (ACI 318 §20.3.2.6, EN 1992-1-1 §5.10.5.2, PTI):

```
P(x) = P0 · exp(−(μ·α(x) + K·x))
```

applied segment by segment walking away from the jack: `P_next = P_prev · exp(−(μ·Δα + K·L_s))`, where L_s is the 3D length of the segment entered and Δα is the full **3D** angle between the two consecutive segments (it includes plan curvature).

Sequential stressing. The stressing operations are applied in order (for a two-end tendon, first end then last end). At each node the force before seating of a later pull is the larger of the new friction curve and the force already locked in by earlier pulls: the jack cannot lower a force.

Anchor seating (wedge draw-in). At lock-off the strand slips back by the seating distance δ (default 6 mm). Friction reverses near the anchor and the force diagram is mirrored about a level p over the slip zone connected to the jack:

- after seating: `F_as(x) = 2p − F_bs(x)` inside the slip zone, unchanged outside;
- p is found so that the strand shortening equals the seating: `2 ∫ (F_bs − p) dx = δ · E · A`.

If the slip zone would be longer than the tendon, the whole tendon relaxes; OpenPT still solves it and warns.

Elongation at a jacking end, per operation (seating already subtracted):

`e = (1/(E·A)) · ∫ (F_bs(x) − S(x)) dx − δ`

where S is the force locked in before this pull. A two-end tendon has one value per end; the total is their sum.

Average force = the length-weighted mean of the force after all operations.

Accuracy. On more than 1,000 tendons of real ADAPT projects, the node forces agree with ADAPT's friction report with a median difference of about 0.06 % and a maximum below 1 %, and elongations per end within about 0.15 % median.

6.6 Long-term losses

Each tendon has the property **Long-term loss** (Stressing tab of the Tendon Editor, the tendon grid, and the **Long-term loss** group on the Tendon tab of **Modify Selection**). It has two choices:

- **Lump sum** (default, as in every earlier version). The force after long-term losses is the force after seating minus the long-term loss of the tendon (Stressing tab, "Long-term loss (MPa)") × the tendon area. This is the value an ADAPT exchange (INP) file carries for each tendon.
- **Calculated.** OpenPT calculates the loss with the method of ACI-ASCE 423 / PTI (Zia, Preston, Scott and Workman, 1979): shrinkage `SH = 8.2e-6·Ksh·Es·(1 − 0.00236·V/S)·(100 − RH)`, elastic shortening (unbonded `ES = 0.5·Es·fcpa/Eci`), creep (unbonded `CR = Kcr·(Es/Ec)·fcpa`, Kcr = 1.6 for post-tensioning) and relaxation `RE = (Kre − J·(SH + CR + ES))·C`, with Kre, J and C from the strand type, fpu and fpi/fpu. The inputs are the **strand type**, the **age at stressing** (default 5 days), E_ci, the **relative humidity** (default 80 %), the **volume-to-surface ratio** V/S, and the **average initial precompression** (default 1.0 MPa); K_cr is 1.6. The method reproduces ADAPT's own loss calculation.

The calculated loss is complete for **unbonded** tendons. For **bonded** tendons the analysis keeps the lump sum and says so in a warning, because the stresses at the tendon at transfer that the method needs are not known to the analysis.

6.7 Equivalent loads of a tendon

A tendon is replaced by the loads it applies to the concrete (load balancing, T.Y. Lin 1963; Aalami, *Post-Tensioned Buildings*). Along the polyline, with segment force $\bar{F}_k$ (the mean of its end forces) and unit direction u_k:

Location	Load on the concrete
start anchor	`f_0 = F̄_0 · u_0`
vertex k	`f_k = F̄_k · u_k − F̄_(k−1) · u_(k−1)`

Location	Load on the concrete
end anchor	$f_n = -\bar{F}_{(n-1)} \cdot u_{(n-1)}$

At a vertex the load is a deviation (curvature) part plus a friction-drag part. The set is exactly self-equilibrated in force and moment. Moved to the slab's reference surface, each load gains a moment $m = (-e \cdot f_y,\ e \cdot f_x,\ 0)$, where e is the height of the strand above (+) or below (−) the reference surface.

Which force is used:

Tendon	Force in the Prestressing load case
Calculated force	force after seating (all stressing operations) minus the long-term loss (lump sum, or the calculated loss of an unbonded tendon) × area
Effective force	effective force per strand × number of strands, constant along the tendon

A calculated-force tendon with no live end is skipped with a warning. A node force below zero is set to zero (warning).

6.8 Prestress loading on the analysis model

When a level with tendons is meshed, OpenPT adds a load case **Prestressing** and, after it, **Dead + Live + self-weight + Prestressing** to the load cases (the **Load:** drop-down, **Analysis Options...**).

How the tendons reach the mesh:

1. Each tendon is built as above. Under a beam, the low point is measured from the **beam soffit**, not the slab soffit.
2. Each tendon segment is cut wherever it crosses a slab-cell edge or the side lines (and end lines) of a beam strip. Each piece gets a **host**: a beam frame if the piece lies in the beam's plan width, else the slab cell that contains it, else the nearest cell within 25.4 mm. Wall shells and columns never host tendons.
3. Pieces with no host (over an opening or off the slab) are **dropped**. The info line reports them: "Prestress: N tendons (S skipped), P pieces, D dropped (L m), residual r". A message such as "Prestress: all 4 tendon(s) of the level were skipped, so no prestress load was applied. Check the tendon geometry." says what to do.
4. Each piece applies +F·u at its start and −F·u at its end. These forces go to the host's nodes with the element's shape functions, together with the moment of the eccentricity between the strand and the node plane. Where two pieces meet on a shared cell edge their loads cancel, so a straight tendon at constant force loads only its anchors, its vertices and the places where its host changes.
5. ADAPT-compatible rules that are on by default: - a piece that crosses a beam at more than 45° to the beam axis loads the slab cells under its ends, not the beam nodes; - the anchor end of a tendon is carried on to the edge of the slab cell that holds it, so the anchor force acts at the cell edge; - a piece inside a beam is shared to the beam nodes by its position on the beam's real axis (this matters where node shift moved the beam); - when an ADAPT project file (.adm) of the same name lies next to the INP, OpenPT reads which tendon segments ADAPT hosted on beams and which on the slab, and follows that choice (it is ignored if the tendon was changed after the .adm was saved).

To see the pieces, switch on **Show tendons** (the check box offered while the picture on show has tendon pieces; the command **Prestress Tendons** has no ribbon button in this build) while a case

containing prestress is shown: shell pieces and beam pieces are drawn in two colours, dropped pieces dashed.

6.9 Hyperstatic (secondary) actions

On a continuous floor the prestress induces support reactions; the actions they cause are the hyperstatic (secondary) actions, which strength design adds to the factored dead and live loads.

Method. OpenPT solves the Prestressing case on the full model and takes the reaction R_n at every supported node. It then releases the supports, keeping only the in-plane stabilisers and the vertical support with the lowest node number, and applies the reactions R_n as loads. The solution of this released model is the **Hyperstatic** load case. Because the loads are the reactions of a self-equilibrated case, the kept supports carry almost nothing.

In the program. When a level has prestress, the Load drop-down lists **Hyperstatic** after Prestressing, and the combination **Strength: 1.2 (Dead + self-weight) + 1.6 Live + 1.0 Hyperstatic** (shown for information). In these pictures the rigid-body tilt of the released floor is removed (a best-fit plane is subtracted), and the title says so. The Hyperstatic case is always available in the load case library and in combinations; the default Strength combinations use it with factor 1.0.

6.10 Tendon display (Tendon Display Manager)

The **Tendon Display Manager** button (Visibility group of the Tendons & Rebar tab) is a toggle:

- click while off: it turns on and opens the dialog **Tendon Display Information**; Cancel turns it off again, OK keeps it on and draws the chosen items;
- click while on: it turns off without opening the dialog; the choices are kept for the next time.

Nothing is drawn unless both the Display Manager and **Show Tendons** are on. The dialog is also on a tendon's right-click menu (**Display Information...**). Choices are kept per user.

Group	Items
Tendon Properties	Tendon ID (`#n`), Tendon label, Area (`Aps=` total area, mm^2), System (`B` bonded / `U` unbonded), Total length (end to end) (`L` straight 3D distance end to end, m), Number of strands (`S=4.00` Decimal as specified, or Rounded Up), Considered or disregarded (`C/D`), Material
Geometry	Control points (default on) with Control point height (CGS above the soffit, mm); Inflection points with their heights; Swerve points (default on, green arrows); Shape (`SH<n>`, n = shape number) with Shape Parameters (`(a,b,c)`); Radii of curvature - Vert (`Rz:` m, then OK in green or NG in red against the tendon's minimum radius); Tendon out of bounds (parts outside every slab region or inside an opening, thick magenta); Analytic Tendons (not available)
Shop Drawings	Chair Heights (see below), Bundle symbol (strand-count symbol: half circle for 1, circle 2, triangle 3, square 4, pentagon 5, hexagon 6 or more), End Anchor Types (`Std Anchor`); Concrete length, Installation length and Length group colors are greyed ("Not available in OpenPT yet.")
XY Plane Curvature	Show Horizontal Curvature Ranges, Show 'OK' range. Spline tendons are coloured by the plan slope: < 1:12 green (OK range only), 1:12–1:8 yellow, 1:8–1:3 orange, steeper red. Fillet tendons by radius: ≥ 6 m green, 3–6 m yellow, 1.5–3 m orange, < 1.5 m red. These mark where bursting reinforcement is needed.

Group	Items
Stressing	Stressing: live, dead ends (arrowhead = live end, short bar across the tendon = dead end; a one-point tendon or an end-on 3D view keeps a small circle); Elongation (`Elong=` mm at each live end); Effective force (`F(eff)=` per strand or `P=` total, kN); Calculated force (`F(avg)=`, `F(min)=`, `F(max)=` kN)

The text height and marker size are those of the Tendon row of the View Items dialog (**View ▸ Display ▸ View Settings**). The property and stressing texts of one tendon are joined with ";" into one label at its first end. **Reset to Default** turns everything off except Control points and Swerve points. If Elongation or Calculated force is ticked, OK runs the friction calculation first when needed.

Tendon heights and tags. View ▸ Layers ▾ ▸ Tendon Heights writes, at each span's high and low points, the height of the tendon (CGS) in mm, and at each live end an anchor tag with the tendon name and count (for example `T7 x 4`). It is on automatically while the Tendons & Rebar tab is open and in the tendon pictures of the reports, until you switch it yourself. **Tab Focus** (see the chapter "Getting started") shows tendons strong while the Tendons & Rebar tab is open and fades them elsewhere. In the plan, tendons are dark red along X, dark blue along Y and dark grey otherwise (**View ▸ Display ▸ Look ▾ ▸ Group Colours** colours them by group); dead ends are drawn as a short bar and live ends keep their arrowhead. Labels never overlap: a label that finds no room is not drawn, and its tooltip still shows it.

6.11 Tendon layout helpers

Two commands of the right-click menu of a tendon (Plan and 3D), after **Trim/Extend Tendon**, help to lay a tendon out. They have no ribbon button.

Low Point to Single Parabola. Sets X2 of the selected tendon spans from their end heights and low-point heights, so that each span is a single parabola. Spans of other shapes are skipped, with a status note. One undo step.

Size Tendon... opens the dialog "Tendon sizing" for one support line. The inputs are: the strand diameter (default 12.7 mm; 15.2 mm is offered too), the force per strand (default from the model, else 120 kN), the covers, and the **target uplift** as a percentage of the dead load per span (default 70 % for the governing span and 60 % for the others). The dialog shows a table per span: force, number of strands, uplift in kN/m and in % of dead load, heights, X2 and warnings. **Apply** writes the force or strand count, the heights and X2 (one undo step); **Cancel** changes nothing.

6.12 Chair heights

Tick **Chair Heights** in the Display Manager to draw, at each chair, a short tick across the tendon and the chair height in mm. The chair height is the underside of the duct above the soffit.

Settings (**Tendons & Rebar ▸ Settings ▸ Chair Settings...**, or **Settings...** beside the box in the Display Manager); one set for all tendons, kept per user:

Option	Default	Rule on OK
Spacing between chairs (m)	1.000 (selected)	below 0.5 m: "Entry is less than minimum value allowed"

Option	Default	Rule on OK
Divisions between high and low points	10	at least 2; odd numbers are raised to the next even number
z-offset (mm)	0	negative becomes 0; capped at the duct radius
Round up/down to the nearest (mm)	5	below 1 becomes 1

Placement per span. One chair at the span's low point and one at each span end (one chair where two spans meet). In spacing mode, chairs are placed from the low point toward each end at the spacing; the last one before the start is dropped if the remainder is under half a spacing (under 0.2 spacing toward the end). In divisions mode, the count applies to **each side** of the low point.

Height. `h = (z − soffit) − r + zterm`, with r = duct height / 2 and `zterm = zOff · (1 − 2·(zmax − z)/(zmax − zmin))` (+zOff at the high point, −zOff at the low point). Heights below r are raised to r, then rounded to the step. At the tendon's first and last points the height is the CGS above the soffit. Where a beam lies under the chair along the span, the beam soffit is used.

6.13 Trim/Extend Tendon

Tendons & Rebar ▸ Modify ▸ Trim/Extend Tendon moves the ends of the **selected** tendons along their end span onto the nearest slab or opening edge, if that edge is within the **Trim Tolerance** (**Tendons & Rebar ▸ Settings ▸ Tendon Settings...**, default 500 mm). Beams, walls and drop panels are not edges. Ends farther away are left alone; an end is never moved past the other end of its span. One press is one undo step; the status line reports "Trimmed/extended n ends of m tendons."

6.14 Elongation Table

Tendons & Rebar ▸ Shop Drawing ▸ Elongation Table... makes the theoretical elongation table for jacking control (bảng trị số giãn dài lý thuyết) from the tendons of the model. It is enabled for an INP document with tendons (single or multi-level), and works while the model is locked.

Data per tendon: Floor, Mark (tendon label), Strands, L (m; the plan length rounded to 0.1 m, or the developed length with the option "Developed length (instead of the plan length)"), Jacking ends n (1 or 2), the table-method values and the real-profile elongation.

Table method (μ, k, α·L) (default):

```
θ  = ROUND(α·L, 3)              P0 = strands·Ap·σp0/1000     σp0 = ROUND(fpu·kcang, 0)
x  = μ·θ/n + k·L/n              wedge = n·6 mm
ΔL = n·σp0·(L/n)·1000/Ep·(1 − e^−x)/x − wedge               rate = ΔL/L
ΔLmin/max = ΔL ∓ tolshort if L < Lshort, else ΔL·(1 ∓ tol)
```

Defaults: fpu 1860 MPa, Ap 140 mm², Ep 195000 MPa, kcang 0.8, μ 0.20, k 0.002 /m, α 0.0324 rad/m, wedge 6 mm per jacking end, tol 7 %, Lshort 15 m, tolshort 15 mm. Where the model has a value (Ap, Ep, fpu, μ, k, seating) it is pre-filled; **Reset to project defaults** restores the defaults above. ROUND is Excel rounding (half away from zero).

Real profile (OpenPT friction): the elongation of the friction calculation above, summed over the jacking operations. It is shown in an extra column in both methods, with the difference in %.

The window "Elongation table" has the parameter box, the **Method:** choice, a grid per floor (STT, Mark, Strands, L, n, θ, x, P0, ΔL, ΔLmin, ΔLmax, rate, ΔL real, Δ%), a **SUMMARY** tab, the options **Group typical floors (a-b)** and **Show on the plan (Elongation text)**, a **Title prefix:** box, and the buttons **Compare with previous table...**, **Export Excel...**, **Export PDF...**, **Copy grid**.

Checks listed under the grid (and written to `_checks.txt`): duplicate marks per floor, strands against the mark's prefix digit (S5-42 = 5 strands), L ≤ 0, a tendon with no jacking end, and a table/real-profile difference above 15 %.

Exports. The Excel and PDF files are written in Vietnamese, whatever the interface language. The Excel workbook has the sheets THONG SO (yellow editable parameters), TONG HOP and one per floor; every result is a live formula referring to THONG SO, so changing μ in Excel recalculates. The real-profile column is a value ("ΔL theo profile thực (OpenPT) (mm)"). The PDF is A4 landscape, 16 rows on page 1 with the notes box, then 25 rows per page, each floor on a new page. The default name is `THÔNG SỐ ĐỘ GIÃN DÀI CÁP ƯST_<yyyymmdd>` in the model folder. **Compare with previous table...** matches rows by floor and mark with an older workbook; changed or new rows are purple in the PDF and orange in Excel, and removed tendons are listed under their floor.

7 Meshing and analysis

7.1 Workflow

1. **Analyse ▸ Meshing ▸ Mesh Generation** – choose the level and the mesh options, press OK. The mesh is made; **View ▸ Layers ▸ Mesh** (a toggle of the group, also a row of **Layers ▾**) draws the cell edges on the Plan and in 3D. The first mesh of a document uses the dialog defaults and says so once in the status line.
2. **Analyse ▸ Run ▸ Run Analysis** (F9) – solves the floor. Choose what to run in **Analyse ▸ Run ▸ Analysis Options...** (below). The progress is in the status bar (see "Long runs").
3. After a run, the result shows in the Plan and 3D. The info line says "Analysed. Pick a result, or run Design." Choose the load case or combination in the **Load:** drop-down and the result in the Results View pane (chapter "Results").
4. **Analyse ▸ Check ▸ Check Model** (F5) opens the **Check Model** window with a summary and the message list; double-clicking a message jumps to the referenced line or item.
5. **Analyse ▸ Run ▸ Run Summary** opens a modeless window with the maxima, the equilibrium check and the solver warnings; it is disabled until a run exists.

A model edit marks the mesh as out of date. On the **Beam** tab, **Mesh and Analyse** meshes with the size in the **Mesh size (mm)** box (when the mesh is missing or stale) and then analyses (see "Beams without a slab").

Analysis Options. Analyse ▸ Run ▸ Analysis Options... opens a dialog with:

- **Model**: **OpenPT mesh**, or **ADAPT deck** to solve the analyser deck that ADAPT wrote in the project's Databases folder (with the deck file and **Browse**); the results of a deck run open in a separate window "ADAPT Deck Results";
- the **tendon** toggle (include the tendons);
- the table **Load Cases to Run**, with the columns **Case**, **Type** (Linear Static, Prestress, Hyperstatic), **Status** (Finished, Not run) and **Action** (**Run** by default, or **Do not run**). Click the Action cell or press space to toggle it; the buttons **Run/Do Not Run Case** and **Run All** do the same for the selected row or for all rows. A case set to "Do not run" is not solved; combinations that use it are not computed and show disabled in the Load drop-down with the tooltip "Not run: <case>". Commands that need every combination (Generate Rebar, Design the Sections, punching, beam rebar, cracked deflection, time-dependent analysis, reports with combinations) refuse while the results were made with a case skipped. The choice is kept per level for the session; a new document starts with all cases set to Run. For a Multi-Level model and for an ADAPT deck the table is shown with the Action column disabled and a line that says that all cases run.

Run Analysis solves the base and the linear derived cases only. Imported modal and response-spectrum cases (of a model made from a SAFE file) are not solved by Run Analysis (F9); they are solved on demand, when one of them is shown or used (the first request solves all of them), which keeps routine static runs fast. The tooltip of the command says so. A model of the older SAFE kind still solves everything.

Support check. At meshing, each connected part of the structure is checked for free rigid-body motion. A part that is free only in the horizontal plane (it could slide or turn about the vertical) gets in-plane stabilisers and the warning "Level 2: the structure is not held in its plane (no support or spring restrains

it ...); in-plane stabilisers added at node ... The net horizontal load of each case goes to them." A part that is free to move up and down, or to tilt, stops the mesh or the run with an error that names the place. A level with no supported part meshes (so that you can inspect it) with one warning, and **Run Analysis** refuses with a clear message. Joints not connected to any element are ignored, with a warning. A static run reports a force imbalance when there is one. The Multi-Level analysis applies the same rules to the assembled model. An imported SAFE model does not get horizontal restraints at the far ends of its columns below the slab.

7.2 Mesh Generation dialog

Field	Default	Meaning
Level	the level shown	the level to mesh
Slab cell size (mm)	1000	suggested size of the slab cells; if the mesher fails, it retries automatically at 0.8 × the size (up to 5 tries) and reports the size used
Mesh pattern	Sparse	**Sparse** or **Uniform** (ADAPT's two slab patterns); Uniform gives a denser mesh near beams, edges and openings
Shift nodes automatically	on	node shift before meshing (see below)
Maximum shift distance (mm)	500	the largest distance a member end may be moved; ADAPT's help suggests 2–3 × the slab thickness

To compare with an ADAPT model, use the same cell size, pattern and shift distance as the ADAPT run.

7.3 How the mesh is built

1. **Slab regions.** A region that lies (at least 98 % of its area) inside a larger one is a **drop panel** or thickened band: its outline becomes a line the mesh must follow, and the cells inside it take its thickness. Where two regions only partly overlap, the smaller region (by plan area) keeps the overlap. Where one region contains another, the one that comes **later in the model order** (the order of the SLAB blocks in the INP) gives the thickness and offset of the cells there.
2. **Openings** are cut out of every region they overlap; overlapping openings are merged. An opening narrower than the 10 mm tolerance is ignored with a warning. An opening that lies outside its region is ignored.
3. **Lines the mesh must follow:** beam centrelines, wall centrelines at the slab, drop-panel outlines and opening outlines. Columns and point supports become fixed mesh nodes.
4. **Quadrilateral mesher.** The slab is meshed into quadrilaterals by recursive loop splitting and smoothing, with the same behaviour as the mesher inside ADAPT-Builder; for the same input it produces the same mesh, so differences in results come from the analysis and not from the mesh.
5. **Cells:** a cell inside an opening is dropped; a distorted or degenerate quad is split into two triangles. Each cell takes the thickness, offset and concrete of the innermost region that contains its centroid. All slab nodes lie on the level's reference plane; the slab's actual position is carried by rigid offsets (offset = −(slab offset + thickness/2)).
6. **Nodes** closer than 10 mm are merged.

7.4 Node shift

Real drawings have member ends that stop slightly short of, or past, the lines they should meet (for example a wall end 150–250 mm from the beam it supports). Imposing such lines gives tiny cells or makes the mesher fail. **Node shift** moves the *analysis* position of member ends and columns onto nearby corners, edges and member lines before meshing; the real position is kept through rigid offsets, so stiffness and load positions stay correct.

The rules follow ADAPT-Builder's own shift, and reproduce its analysis positions for nearly all member ends of the tested projects:

- Every move is shorter than the **Maximum shift distance**.
- Opening corners snap to slab-region corners or edges first.
- Targets are slab and opening corners and edges, drop-panel edges and corners, other members' ends and lines, and columns. A beam end goes to a column beside it (the column does not move to the beam).
- A beam running along an axis-parallel slab edge goes onto the edge line.
- Members are not straightened: a member may stay slightly skewed.
- Of extending/trimming along the member and moving perpendicular to an edge, the shorter move wins; an end already on a corner or edge stays there.
- Columns snap to the nearest corner or edge target.

7.5 Members in the analysis model

Beams. Frame elements along the mesh edges on the beam line. Section: A = width × full depth (the slab under the beam is not deducted), I about both axes, J = I1 + I2. The beam is offset down to its own mid-depth from its top. A beam that runs off the slab to a column gets a frame to the column. Beams along a notch of the slab outline and free beams are modelled as frames as ADAPT does.

Columns. Two frames above and two below the slab (storey heights from the level table; 3000 mm if there is no level above or below). Rectangular columns use w × d; circular columns use an inscribed 20-gon. Far ends: a column below is fixed against vertical movement and rotations at its base (ux, uy free); a column above is fixed in rotation at its top. **Under a deep beam** (when the beam soffit is at or below the column's mid-height), there is no mid-height node: the column is a half-height frame tied rigidly to the slab node, as in ADAPT.

Walls. ADAPT's "Simplified" method: a structured grid of shells, two rows above and two rows below the slab, between the slab-side row of nodes on the wall line and the far end. The slab-side row is thinned (every other point removed) while the cells are much shorter than the cell size; over openings or where the wall leaves the slab, points are added at a regular step. Far ends are restrained like columns. A wall below the slab whose end lies under a deep beam is tied to the slab node by a rigid arm instead of having a mid-height node.

In-plane stabilisers. To prevent in-plane rigid-body motion, the slab is held in plan at the first two vertices of the first slab region of the file (ux and uy at the first, one direction at the second), the same choice as ADAPT. This matters for the prestress case, where the in-plane forces are large. A floor that nothing holds in its plane gets stabilisers and a warning (see "Support check").

7.6 Loads on the mesh

- **Area (patch) loads** are clipped by every slab cell; each piece's resultant is placed at its centroid and shared to the cell's nodes.
- **Line loads** along a beam go to the beam's nodes; elsewhere they are clipped by the cells.
- **Point loads** go to a beam that contains the point, else to the cell.
- **Self-weight** is not a nodal load: the solver applies it from the concrete unit weight of every element (load case "Self-weight").
- Parts of loads that lie off the slab are **not applied**. A warning is given per load case when more than 0.1 % (or 0.1 kN) is lost: "Load case <name>: <dropped> kN of <defined> kN lies off the slab and is not applied (<n> loads)", and loads more than 50 % off the slab are named.

Load cases: every user case (Dead load, Live load, ...), **Self-weight**, **Dead + Live + self-weight**, and with tendons **Prestressing**, **Dead + Live + self-weight + Prestressing**, **Hyperstatic** and the Strength combination.

7.7 The finite-element solver

Item	Formulation
Nodes	6 degrees of freedom
Slab and wall shells, bending	quadrilateral Q19 (Clough–Felippa), Kirchhoff plate (no transverse shear deformation); triangles from the HCT element
Shells, membrane	QM6 (Taylor–Beresford–Wilson) quadrilateral; constant-strain triangle
Shells, drilling rotation	ADAPT's drilling stiffness rule, taken from the bending stiffness
Beams and columns	3D Euler–Bernoulli frames
Offsets	rigid links on both element types
Self-weight	from unit weight × thickness (shells) or area (frames), applied through the offsets
Cracking	per-element stiffness ratios in two directions (used by Cracked Deflection)
Equation solver	sparse direct solver that does not need a positive-definite matrix

After every solve the program checks equilibrium (applied loads + reactions = 0, in force and moment) and reports it, with any degree of freedom that no element touches ("Freedoms not touched by any element (held at zero)").

Agreement. On eleven project decks the displacements of OpenPT's solver are checked against ADAPT's analyser on the same model, with a test tolerance of 1 %. The SAFE-specific drilling formulation is used only for SAFE F2K models; INP models use the ADAPT-compatible element.

7.8 Speed

For an analysis, the stiffness matrix is assembled and factorised once and reused for all load cases, in a single-level and in a Multi-Level run; only the Hyperstatic case (a different support set) and the cracked run need their own. Results are identical to the earlier one-solve-per-case approach. The solves of the SAFE-type models run in parallel. When memory is short, the overlap of two factorisations is refused and the memory check says so (see "Long runs, stability and recovery"). A long run shows its progress in the status bar and can be cancelled.

7.9 Modal analysis

Analyse ▸ Modal ▸ Modal Analysis Options... and **Run Modal Analysis** find the natural periods and mode shapes of the model. The run can be cancelled and shows its progress; it never changes the static results.

Options (stored per model):

Option	Default	Meaning
Number of modes	3	1 to 100
Mass source	Self weight 1.0, superimposed dead 1.0, live 0.1, others 0	Factor of each load pattern that contributes mass; prestress and hyperstatic patterns cannot be listed
Mass directions	Vertical (Z) only	or **Vertical and lateral (X, Y, Z)**
Stiffness	Uncracked (gross)	or **Cracked: from the cracked-deflection combination** (offered only after a cracked deflection run)

Results. A **Modal** tab shows the table of modes: mode, period T, frequency f, ω, eigenvalue and the mass participation UZ in % with its cumulative value (and UX, UY when lateral mass is on); it can be exported as CSV. The Results View tree has the leaf **Modes ▸ Mode 1 ... n**: a contour of the normalised amplitude in the Plan and the deformed shape in 3D, with the status line "Mode 3: T = ..., f = ..., UZ ...%". Raising the number of modes marks the old modal results out of date ("Run Modal Analysis for 8 modes"). A model made from a SAFE file solves its imported modes and response-spectrum cases from the imported modal data (see the chapter "Working with SAFE models").

The command line `--modal` makes the same run (see "Command-line use").

7.10 Time-dependent analysis

See "Time-Dependent Analysis (ACI 209)" in the chapter "Design".

8 Loads, criteria and combinations

8.1 Load tools (Loads ▸ Add Loads)

Button	Use
Add Point Load	click the position; the command repeats
Add Line Load	click the first and second point; the arrow offers **Continuous Modeling** (the end of one load starts the next)
Add Patch Load	click the corners; Enter, C, End, a double-click or right-click **Close/End/Accept** closes it (3 corners or more)
Line Load Wizard	dialog "Create Line Load Automatically": line loads along every edge of the selected slab or opening, or along the axis of the selected beam or wall
Patch Load Wizard	dialog "Create Patch Load Automatically": one patch load over each selected slab

The wizards need a selected Slab, Opening, Beam or Wall; each asks for the **Load case** and the value (kN/m or kN/m², default 0; 0 creates nothing) and reports "N loads were generated successfully." A patch load from the wizard covers the slab outline; openings are not subtracted, but load over an opening is not applied by the mesh.

New loads are created with all values 0 and without a dialog. Set the values before placing with **Modify ▸ Item Properties** while the command is active (the dialog title then ends with "(next load)"), or double-click the load afterwards. The command ends with Esc, typing x, or right-click **Exit**. If the model has no user load case, Dead load and Live load are created first.

Load dialog ("Point Load", "Line Load", "Patch Load"), always in SI units:

Tab	Point	Line	Patch
General	#, Label, Load case, Fz [kN], Mxx, Myy [kN·m]	Fz1 [kN/m], Variable Magnitude, Fz2, Mx1, My1, Mx2, My2 [kN·m/m]	Fz1 [kN/m²], Variable Magnitude, Fz2, Fz3
Loads	Fx, Fy [kN]	Fx1, Fx2, Fy1, Fy2 [kN/m]	Fx, Fy [kN/m²] and Mx, My [kN·m/m²] at vertices 1–3
Location	Vertical start offset [mm]	start and end offsets, the points	offset, the points

Signs: **a positive Fz is downward**; moments follow the global axes; the vertical offset is positive downward. For a variable patch load, values 1, 2 and 3 belong to the first three corners. In the plan, loads are drawn in teal while the Loads tab is open (Tab Focus). The labels read value and unit with three significant figures: `Fz = 14.2 kN`, `w = 12.0 kN/m`, `q = 5.00 kPa`, `Mx = 3.50 kN·m`, in the display units. The **Show Loads** group (**Loads ▸ Show Loads**) shows or hides each kind (**Show Loads**, **Show Point Loads**, **Show Line Loads**, **Show Area Loads**), and **View Load Settings** sets symbol size and texts. In 3D a load is drawn as short arrows standing on the slab with a thin outline of its area; an area (patch) load has a closed lid joining the tails of its arrows, so that it reads as one load.

8.2 Load cases (Loads ▸ Define ▸ Load Cases)

Loads ▸ Define also has **Load Patterns** (a read-only list of the load patterns, for a document of the older SAFE kind), **Load Combinations** and **TCVN deflection...**. The **Load Case Library** window:

- **Reserved Load Cases:** Dead load and Live load (tick to include), and Selfweight, Prestressing and Hyperstatic, which are always present and cannot be changed.
- **General Loads (Gravity):** the user cases, with **Label**, **Add** and **Delete**. Names may use letters, digits, spaces, "" *and* "-"*; reserved names and names starting with* "*Lateral*" are refused.
- Unchecking or deleting a case that has loads asks for confirmation and deletes its loads. A case used by a combination cannot be deleted.

8.3 Design criteria and design codes

Design ▸ Criteria ▸ Design Code... opens the **Criteria** window on its **Design Code** tab. It is the **one place to choose the design code of the document**. The code drives the load combinations, the flexure, shear, punching and deflection design on every kind of model (a model of your own, or one made from an ADAPT or a SAFE file).

Code	Notes
TCVN 5574:2018	Vietnamese code; the default for a model made from a SAFE file. See the chapter "TCVN 5574:2018 design".
EC2 (EN 1992-1-1)	The default for a new model and for an INP model.
ACI 318-14 ("ACI 2014/IBC 2015")	
ACI 318-19 ("ACI 2019/IBC 2021")	Follows ACI 318-14, except that two-way and one-way shear use the size effect λs = √(2/(1 + 0.004 d)) ≤ 1 (22.6.5.2 and Table 22.5.5.1; for slabs and, for beams without stirrups, in the beam shear check) and that the minimum flexural steel of RC slabs is 0.0018 A_g (8.6.1.1).
SP 63.13330.2018	Russian code; runs on the TCVN engine with its own rebar table (short-term R_sc), the TCVN combination rows (the names stay `ULS_TCVN`) and the punching and shear rules. Rebar strengths follow Amendment 1 (A400 R_s 340 and R_sc 340; A500 435 and 400; B500 415 and 380). Prestressed rules are those of TCVN. Columns, torsion and bearing design as a model design are not in the product (they are in Section Check). The status bar and the reports show "Code: SP 63.13330.2018".

Older ACI editions (2005, 2008, 2011) are read from files that store them but are not offered in the list. A file with no stored code opens with TCVN (a model imported from SAFE) or EC2 (an INP); an unknown value opens with the default and one message. **File ▸ New model...** has no code row: a new model takes the code last used. Switching a document between TCVN and SP 63 converts the rebar grade ("The rebar grade CB400-V becomes A400.") and back. Changing the code asks first; it recomputes Ec of every concrete and replaces the code's default combinations.

The window also has the crack-width options (Report probable crack width, Limit probable crack width, Crack width limit), the material factors, the strength reduction factors, and, in the group "TCVN materials" / "SP 63 materials", the **Humidity (%)** drop-down (>75, 40-75 default, <40) that sets the creep coefficient φ_b,cr; beam deflection under TCVN and SP 63 uses the T-section flange and this humidity. A change is one undo step. The **Combinations for design** choice has two entries: **From the model** (default: the combinations of the document) and **Generated for the design code**. The Convert

dialog of a SAFE import (default options, no dialog) preselects the document code. `--code` of the command line accepts TCVN.

	EC2	ACI
γc / γp / γs	1.5 / 1.15 / 1.15	1 / 1 / 1
φ: bending initial, bending, shear one-way, two-way	1, 1, 1, 1	0.9, 0.9, 0.75, 0.75
Ec from fc	`22000·((fc+8)/10)^0.3` (EN 1992-1-1 Table 3.1; fck 40 gives 35220 MPa)	`0.042747·wc^1.5·√fc`

The EC2 modulus applies to a new material and when you edit f_c; a model file that stores E_c keeps it. ACI 318-02, 08 and 11 (ACI02, ACI08, ACI12, read from old files) give two-way slabs with only bonded tendons a minimum rebar of 0 ("bonded tendons: no minimum rebar (ACI 318-11 18.9)"); ACI 318-14 is unchanged.

Allowable Stresses (**Design** ▸ **Criteria** ▸ **Allowable Stresses**). EC2 defaults: initial tension 0.30·fci^(2/3) (EN 1992-1-1 f_ct, top and bottom); final tension 0.30·fck^(2/3); rebar 0.80·fyk; tendons 0.75·fpk; compression 0.45·fck (quasi-permanent), 0.60·fck (frequent), 0.60·fck,i (initial); crack width 0.30 mm (RC), 0.20 mm (PT frequent, bonded; PT quasi-permanent, unbonded). ACI defaults: tension 0.5·√f'c (sustained and total), compression 0.45·f'c (sustained) and 0.60·f'c (total), initial 0.25·√f'ci and 0.60·f'ci; crack width limits 0.30 / 0.20 mm.

Materials (**Design** ▸ **Criteria** ▸ **Materials**, with the choices **Concrete**, **Rebar**, **Prestressing**): one window per kind with a list, **Add** and **Delete**:

- Concrete: Label, Wc [kg/m³] with Normal / Semi light weight, unit price, cylinder fck or cube fck,cu (fck = 0.8·fck,cu), Ec (recomputed when fck changes; may be typed over), initial strength ratio (default 0.75), Poisson's ratio, aggregate size, thermal coefficient. New concrete: fck 28, Wc 2400, ν 0.20.
- Rebar: fy, Es, price. New: fy 460, Es 200000.
- Prestressing: fpu, fpy, Eps, price. New: fpu 1860, fpy 1700, Eps 200000.
- A material in use, or the last of its kind, cannot be deleted.
- **Material grades.** Under TCVN (and SP 63) every list offers the TCVN concrete grades B15 to B60 and the rebar grades CB240-T, CB300-T, CB300-V, CB400-V and CB500-V; **Custom** opens a dialog for typed R_b, R_bt, R_b,ser, R_bt,ser, E_b and φ_b,cr (concrete) or R_s, R_sc, R_sw and E_s (rebar). The EC2 concrete dialog has a grade list C12/15 to C50/60. ACI offers f'c from 3000 to 10000 psi. The punching dialog has no custom material.

The design code, allowable stresses and the values the INP cannot hold are stored in the model's criteria sidecar (`<inp>.criteria.json`); materials and load cases are written to the INP.

8.4 Load combinations (Loads ▸ Define ▸ Load Combinations)

Loads ▸ **Define** ▸ **Load Combinations** (on the Beam tab: **Beam** ▸ **Loads** ▸ **Load Combinations**) opens **Define Load Combinations**:

- Buttons: **Add New Combo...**, **Add Copy of Combo...**, **Modify/Show Combo...**, **Delete Combo**, **Add Default Design Combos...** (adds the code's default combinations that the list does not have yet).

- **Load Combination Data** form: **Name** (unique among combinations and cases), **Type**, **Notes**, a grid of Load Case/Combo Name × Scale Factor (with Add, Modify, Delete), and **Design use**: **Strength (ULS)**, **Service (SLS)**, **Deflection**.
- Types: **Linear Add**, **Envelope** (max and min), **Absolute Add**, **SRSS**, **Range Add** (sum of positive parts and of negative parts). A combination may contain other combinations; a cycle is refused.

No re-analysis. A combination's results are built from the stored results of its members (displacements, reactions, element forces, section actions). After OK the Results lists update at once. Design results that depend on combinations (strip design, punching, cracked deflection, calculated rebar) are marked out of date and must be re-run; no new analysis is needed. A change that alters a combination also drops the cracked deflection of the level, clears its picture and says on the info line to run it again; saved cracked results made with other combinations are reported out of date on opening. The "combinations changed" note clears after the design and the shear check are run again. Combinations can be edited while the model is locked.

Storage. Linear Add combinations of load cases are written to the INP's LOAD_COMBINATIONS block on Save, in ADAPT's layout, so ADAPT sees them. The design option written follows the Design use: Strength → STRENGTH, Service → Service Frequent (EC2) or Service Total Load (ACI), Deflection → Cracked Deflection; an imported combination keeps its own option (for example Quasi-Permanent, Initial or Long-Term) while its Design use is unchanged. Types and nesting the INP cannot carry are kept in `<inp>.combos.json` and marked "OpenPT only".

Default combinations. When the INP lacks them, OpenPT lists the code's defaults as "(default)", read only:

EC2 label	Design option	Factors
Frequent	Service Frequent	Selfweight 1, Dead 1, Live 0.5, Prestressing 1
Strength	Strength	Selfweight 1.35, Dead 1.35, Live 1.5, Hyperstatic 1
Service(quasi-permanent)	Service Quasi-Permanent	Selfweight 1, Dead 1, Live 0.3, Prestressing 1
Initial	Initial	Selfweight 1, Prestressing 1.15
Sustained_Load	Cracked Deflection	Selfweight 1, Dead 1, Live 0.3, Prestressing 1
Long_Term	Long-Term Deflection	Sustained_Load 3

ACI: Service(Total Load) 1/1/1/1; Service(Sustained Load) Live 0.3; Strength(Dead Load Only) 1.4 D + 1.0 Hyp; Strength(Dead and Live) 1.2 D + 1.6 L + 1.0 Hyp; Initial; Sustained_Load; Long_Term.

A Cracked Deflection combination may not contain Hyperstatic (use Prestressing instead).

SP 63. The default rows of SP 63 use the live load factor 1.2 when every live area load is 2.0 kPa or more (SP 20.13330.2016 8.2.7); the row's clause text names it and the status line says "SP 63 live load factor 1.2 ... (lowest 2 kPa)". With an Initial combination, TCVN and SP 63 strip design gets a transfer check `TCVN_Transfer` ("R_bp ... MPa"); an older document has no Initial row until you add it with **Add Default Design Combos...**.

8.5 TCVN total deflection

Loads ▸ Define ▸ TCVN deflection... builds the TCVN total-deflection combinations `F = f1 − f2 + f3` (TCVN 5574:2018 / TCVN 2737:2023) on the open model (entries named `TCVN_F`, `TCVN_f1`, `TCVN_f2`,

`TCVN_f3`; they have no design use of their own). It is described in the chapter "TCVN 5574:2018 design", section 1. Because the formula is linear in the load cases, you can also build the same result by hand:

- `f1 = ks·r·(G + Q)`, `f2 = ks·r·(G + η·Q)`, `f3 = (1 + ϕ)·r·(G + η·Q)`, with ks = 1/0.85, η = long-term part of the live load (default 0.35), φ = creep coefficient of the concrete class and humidity, r = E_model/E_b.
- So `F = a_G·G + a_Q·Q` with `a_G = r·(1 + ϕ)` and `a_Q = r·(ks·(1 − η) + η·(1 + ϕ))`.

Create a **Linear Add** combination in **Loads ▸ Define ▸ Load Combinations** with factor a_G on Selfweight and Dead load and a_Q on Live load, and show its Slab deflection. This is an uncracked deflection (no Prestressing term unless you add it).

9 Results

This chapter describes how results are shown: the Results View pane, the contours with their colour bar, bands, labels and maximum and minimum markers, the diagrams, the same results in 3D, and the section actions behind the design.

9.1 Showing a result

After a run, the result opens by itself in the drawing area (Plan and 3D, with the deformed shape in 3D), and the **Results View** pane opens once per finished run (you may close it).

- **Landing result.** If you have not picked a result, the Plan opens on the **deflection contour of the first service combination** (Frequent first, then Sustained, else the first combination); the **Load:** box shows it. When the model has strips and design sections, the **strip moment** diagram is the default result instead (see "Strip moment diagrams"). While a result shows, the mesh, the rebar callouts and the loads are hidden; they come back when you leave the Results view. The info line says "Analysed. Pick a result, or run Design." after a run and "Designed. Pick a result or open the Report." after Design.
- **Analyse ▸ Results ▸ Results Display Settings** (a toggle; the Results group of an INP-kind model holds only this button and the three results-scale buttons) shows or hides the docked Results View pane at the right. It needs a mesh ("Run Mesh Generation first"). The pane is stacked below the Properties pane when both show.
- **Load:** drop-down, at the end of the document tabs: the groups **Load Combos** and then **Load Cases** (group headers are not pickable). Picking one sets the load shown in the Plan and in 3D; the title of the pane reads "Results View: <load>". The drop-down is hidden until there is a load to pick and reads "(no results)" until results exist. A combination that uses a case set to "Do not run" is disabled with the tooltip "Not run: <case>".
- **Plan + 3D (results stay where you asked).** With the Plan and the 3D view side by side (**View ▸ Views ▸ View ▸ Plan + 3D**), a result is drawn only in the pane that was **active when you asked for it**: the contour, diagram, deformed shape, labels, colour bar, maximum and minimum markers and the info card. The other pane shows the clean model. "Asking" is a command of yours that changes the shown result: ticking or choosing a result, contour, diagram, deformed shape or result type, or choosing another load case or combination of a result that shows. Clicking into the other pane, or rotating it only moves the accent line; the results stay where they are. To move the results to the other pane, make that pane active and ask again (tick or choose the result again). When you turn Plan + 3D on, the results are in the Plan. Results asked while the 3D pane is active show in 3D only.
- A model edit marks the results out of date ("Results out of date — press Apply").

9.2 Results View pane

At the top of the pane: **Apply** (refresh: re-solve if out of date, then redraw) and **Clear All** (unticks every result; the load selection stays). **A tick takes effect at once**; Apply stays as a refresh. The pane has no progress line or Cancel button of its own (the run bar is in the status bar), and no **Expand all** / **Collapse all**. The pane has the tabs **Analysis** and **Display**, and, for a model made from a SAFE file, a **Report** tab (see "Working with SAFE models"). The load choice is the **Load:** drop-down of the document tabs.

Analysis tab. Each family has a **single choice**: at most one contour and one diagram can be ticked at a time. The **Beam** diagram and the **Design Sections** diagram exclude each other; the **Punching Shear** leaves stay independent, except that **Stress check** and **Condition** (which colour the perimeter) exclude each other.

Group	Leaf	Unit and sign
Slab ▸ Deformation	Slab deflection (contour)	mm
Beam ▸ Actions (Diagram)	Beam axial force (+ tension), Beam lateral shear (in-plane), Beam shear (vertical; + left part pushed up), Beam moment (vertical bending; + sagging), Beam lateral moment, Beam torsion	kN, kN·m
Design Sections ▸ Deformation	Deflection at the section reference point	mm
Design Sections ▸ Actions	Bending (+ compression on top, sagging; the strip moment), Axial (+ compression), Shear, Shear in-plane, Bending normal to plane, Torsion	kN, kN·m
Design Sections ▸ Stresses	Top, Bottom (+ tension), P/A (FEM) (+ compression)	N/mm^2
Design Sections ▸ Reinforcement (Longitudinal)	Calculated (required), Calculated (provided), Base	mm^2
Punching Shear	Stress check, Stress ratio, Condition, Reinforcement, Reinforcement layout	—
Modes ▸ Mode 1 ... n	Normalised amplitude (after a modal run)	—
Cracked results	Crack ratio, after Cracked Deflection	—

The beam leaves carry the ADAPT name in the tooltip; in Vietnamese mode every leaf is translated. A beam on a level with no slab also has a **Beam deflection** diagram (see "Beams without a slab"). A result title reads "Beam moment (kN·m)", without a group word in front. A result that needs design sections when there are none shows the offer "No design sections yet. Generate them now?" with a **Generate Sections** link (the same command as **Design ▸ Section Design ▸ Generate Sections**), and a strip result with no strips shows "No design strips yet. Create strips and sections now?" with the link **Create Strips and Sections** (it runs **Auto Strips** in both directions with its defaults, then **Generate Sections**, as one undo step, and the strip moments show). Quantities that have nothing to draw in the model are not offered.

Punching Shear ▸ Reinforcement layout needs a punching result. At each support with shear reinforcement it draws the legs, the outer contour and a label in the Plan and 3D views; for EC2, ACI and SP 63 stirrups it draws every stirrup row (see "Punching shear").

Section stresses: `P/A = Axial/A`, `Top = −Axial/A − Bending·(z_top − ȳ)/I`, `Bottom = −Axial/A + Bending·(ȳ − z_bot)/I` (tension positive).

For an envelope, the slab contour shows the largest downward displacement per node ("Envelope (min Z)"); diagrams show the max and min curves.

9.3 Contours

Every contour (the Plan, the 3D deformed shape, the crack ratio, the modal shape and, on the older SAFE kind, the F2K results) uses the same presentation:

- **Colour bar.** At the top right of the drawing area, the colour bar names the **quantity and its unit** (it always names the contoured quantity) and carries a sub-line with the load case. A beam diagram drawn over a contour adds a line under the title, for example "Beam moment (kN·m): max 833, min −249". It shows the same colour **bands** as the drawing: blocks of the bands, with their edge values as ticks, round tick values, and the maximum and minimum. On the screen (Plan, 3D and the analysis view) the bar has **no card**: there is no opaque panel, its texts have a thin halo in the background colour so that they read over empty background, and the bar is painted **behind the model**, so a slab, beam, column, wall or contour that overlaps it is drawn over it and no structural element is hidden. Zoom Extents still leaves room so that the bar sits beside the slab. Pictures and reports (the report images, `--render-*`) keep the bar as before, with its white card on top. The Plan legend card has no contour swatches.
- **Bands.** A contour is drawn as clean colour bands: 8 to 16 bands of round steps (1, 2, 2.5 or 5 × 10^k) with thin boundary lines. The crack ratio is 5 bands of 0.2. The bands are 85 % opaque by default (**Fill opacity**).
- **Colours.** Ranges that cross zero (moments, stresses) use a diverging blue-white-red scale, and one-sign ranges (deflection) a sequential blue ramp (the **Display** tab choice **Contour colours: Engineering** (default) or **Rainbow**).
- **Smooth or banded. Smooth contours** (Display tab, **on by default**; saved in `plan.ini`) makes the bands follow a smooth field resampled on a raster over the slab, instead of the element facets; it is built off the user-interface thread, and the exact bands show until it is ready. Values and the maximum and minimum are unchanged. With the option off, the bands follow the element facets (banded contours).
- **Contour value labels. Contour labels** (check box, off by default) and **Label every** (1 to 10, default 2) write the edge value of the band on every N-th contour line of the Plan, turned along the line and upright, with a white halo. Both act at once. The labels use the **Decimals** of the tab.
- **Decimals** (spinner, default 2) sets the decimals of the colour bar, the maximum and minimum texts and the labels; **Fill opacity** sets the opacity of the contour fill.
- **Maximum and minimum markers.** The largest and smallest values are marked on the slab and labelled "max 0.410 mm" and "min -13.5 mm" (three significant figures and no "+"; **Display Max/Min**, on by default; the marker sits on a slab node). The labels avoid the colour bar and each other.
- **The model under a contour.** While a contour, a diagram or a deformed shape shows, the structure is drawn as thin grey lines (no fills, no family colours); rebar, loads, labels and strips are not drawn and tendons are drawn at 25 % in grey. **View ▸ Display ▸ Look ▾ ▸ Show Details** (off by default) brings them back. Columns and walls stay filled grey over a contour, and drop panels are drawn as outlines.

Results in 3D. The 3D tab draws the deformed slab as a coloured, shaded surface (switching **Shaded** off keeps the lines), with the same colour bar, bands and **Display** tab settings as the Plan: **Contour labels** and **Label every** (the labels are turned along the projected line and are drawn only where the face looks at the camera), **Decimals**, **Fill opacity** and **Display Max/Min** (the maximum and minimum markers in 3D). The settings act while only the 3D view shows a contour, also in a Multi-Level model. With a **diagram** result in 3D (moment, force, shear and the like), the structure under it is drawn **shaded**, as in the plain 3D view (columns, walls and drop panels are solid, not wire boxes), at about 60 % opacity and

in the same colours, so that the diagram and its labels stay on top and readable. Walls and columns are grey-shaded in the 3D result view, and the 3D bands are smooth and lit. A contour or the deformed shape keeps its own look.

9.4 Strip moment diagrams

Where the model has strips and design sections, the **strip moment diagram** is drawn along each support line: filled light blue for sagging and light red for hogging, the largest ordinate about 1/6 of the strip width, labels at the span peaks and at the supports (in kN·m), and a legend chip "Strip moment, <case> (kN·m)". After **Run Analysis** with strips and sections the strip moment is the default result. The same diagram is the **Design Sections ▸ Actions ▸ Bending** leaf; to show it, tick that leaf in the Results View pane (the command **Display Strip Results**, which opened the pane on it, has no ribbon button in this build).

9.5 Diagrams

Strip and beam diagrams are drawn in the Plan and 3D views with one common scale: ordinate = value × 2.8 m / M × f, where M is the largest absolute value of that result over all drawn sections and f the user factor. **Results Scale Up**, **Results Scale Down** and **Default Results Scale** (**Analyse ▸ Results**; the buttons read Scale up, Scale down and Reset scale) multiply f by 1.1 / 0.9 or reset it. **Numerical Display**, **Display Max/Min** and **Perpendicular Projection** are rows of the **Display** tab of the Results View pane (see below); they have no ribbon button.

Labels of diagrams are a few meaningful numbers: the member ends at supports or at jumps, and the peak in each span; neighbours of nearly equal value are dropped. In 3D the labels are projected to the screen, larger magnitudes first, overlaps dropped (zooming in brings more back).

9.6 Display tab

The **Display** tab acts on what it says; a row takes effect at once (no Apply).

Row	Default	Acts on
Contour colours	Engineering (or Rainbow)	Contour colours
Smooth contours	On	Contours, Plan and 3D
Contour labels, Label every	Off, 2 (1 to 10)	Contour lines
Decimals	2	Colour bar, max and min texts, labels
Fill opacity	slider, percent	Contour fill
Results scale factor	1.00	Diagrams
Numerical Display	Off	Diagrams
Display Max/Min	On	Diagram labels; max and min markers of contours
Perpendicular Projection	Off	Diagrams

Row	Default	Acts on
Fill colours (positive, negative), envelope line colours (max green, min blue)	blue, red, green, blue	Diagrams

Colours are swatch buttons with a palette of 16 colours and **More...**. While only a contour shows, the rows that act only on diagrams (results scale factor, Numerical Display, Display Max/Min for diagrams, Perpendicular Projection, the fill colours and the envelope line colours) are greyed with the tooltip "Acts on the strip and beam diagrams".

Units on screen are `kN·m`, `kN·m/m`, `kN`, `kN/m`, `MPa` and `mm`; the CSV files and the table headers of reports keep `kN.m`.

9.7 Section actions

For every design section and every load case or combination, OpenPT integrates six actions: Axial, Lateral (in-plane shear), Vertical shear, Torsion, Bending and Mz. It uses the same **free-body method** as ADAPT-Builder's "Design the Sections":

1. The element nodal forces `K_e·u_e` of every slab shell and beam frame that the cut crosses (within 5 mm) are taken in global axes.
2. Shell forces are scaled by the fraction of the element's chord that lies on the cut.
3. Only the nodes on one side of the directed cut are used.
4. The forces and moments of those nodes are summed about the section centroid (from the cut polygon, at its true elevation).

Bending > 0 is sagging and Axial > 0 is compression. Every beam frame the cut crosses is included. Beam diagrams use the frame end forces along each beam.

Agreement. With ADAPT's own element forces, this integration reproduces ADAPT's section actions (N, M and V within 2 %) on about 88–97 % of section and combination rows of five real projects; the remaining differences sit where a node lies within 5 mm of the cut or the cut passes an opening. Differences on a full OpenPT run therefore come from the analysis model (mesh, node shift, prestress loads), not from the integration. Twisting moments (Wood–Armer) are not included in the design moment.

9.8 Section Cut (View ▸ Section)

Section Cut draws a cross-section of the structure along a line, flat in the plan.

1. **View ▸ Section ▸ Create Section Cut**.
2. "Enter the First Point of the Line." – click.
3. "Enter the Second Point of the Line." – click (two equal points are refused).
4. "Select an insertion point:" – a dashed preview follows the cursor; click to place it.

The cut is created and selected ("Section N created."); one cut per command; Esc cancels. The drawing is not stored as lines: it is rebuilt from the current model every time, so it follows later edits. It shows, 1:1 in mm, the slabs, drop panels, beams, columns and walls crossed, with the caption "Section N". Cuts are stored with the strips in `<inp>.strips.json`, not in the INP. **Section Cuts** (**View ▸ Section ▸ Section Cuts**, a small toggle) shows or hides all cuts.

Reinforcement in the cut. The drawing also shows:

Item	Mark
Tendon crossing the cut	dark red filled circle at the CGS, equivalent diameter of the total steel area
Bar crossing the cut	filled circle at the bar centroid, bar diameter (banded: d·√n)
Distributed bar	one circle per bar line that crosses
Bar lying in the cut plane	a line over the overlap
Mesh	one circle per bar along the part of the cut inside the mesh; a line when the bars are parallel to the cut

Colours are those of the Rebar Display Manager (top, bottom, top mesh, bottom mesh). The marks follow Show Tendons, Show Rebar and the Display Manager filters.

9.9 Design section information panel

Right-click a design section ▸ **Properties...** (or double-click it) opens the Support Line dialog on its **Design Sections** tab:

1. **Current Design Section** – the section combo; changing it selects that section in the plan.
2. **Geometry:** `L =` cut length (L trib + R trib), `H =` full depth, `Beff =`; **Details...** shows end coordinates, area, inertia, centroid and fibre distances ("Design Section Details").
3. Two pictures at a common scale: **Physical section with base rebar and tendons**, and **Design section (idealized) with calculated rebar** (squares sized by area: green top, red bottom).
4. **Reinforcement:** `Base rebar: As = … mm2 Rho = … %` (Rho counts base + calculated steel over the gross area), `Tendons: Asp = … mm2, Calculated: (envelope) Astop = … Asbot = …` with the governing combination, or "not designed". **Details...** lists each item with its area and its CGS from a chosen **Reference Line, RL:** (Top / Bottom / Centroid).
5. **Design summary:** `Criteria:` (Two-Way / One-Way / Beam, PT or RC), `Moment capacity M+ / M-` (strength method of the strip design, with the provided steel), `Demand (strength) M+ / M-`, `D/C ratio` (red above 1; "Exceeds" when the capacity is 0). The shear capacity lines are left blank.

How the cut is formed. Over each interval of the cut, the section is the union of the slab region there (the smallest region by area where regions overlap), of the beams whose outline contains the interval (each from its own top, using the beam's offset, down by its depth), and of the drop panels (from their own top). A beam whose axis is within 30° of the cut adds nothing. A beam stem enters a support line's sections only if the line "detects" the beam: a vertex of the line lies inside the beam outline (5 mm tolerance) and the line segment from that vertex is within 30° of the beam axis; detected beams enter every section of that line. Tendons that cross or touch the cut (within about 1 mm) are included.

10 Design

10.1 Design strips: support lines, tributaries and sections

Design is done on **design strips**, as in ADAPT-Builder. The tools are in **Design ▸ Strip Modelling**, **Design ▸ Strip Layers** and **Design ▸ Section Design**; the design code is chosen in **Design ▸ Criteria ▸ Design Code...** (chapter "Loads, criteria and combinations"). The same strips serve every kind of model, also a model made from a SAFE file.

Concepts.

- **Support line:** a polyline with a direction X or Y (the direction of the reinforcement it designs). Its vertices are the supports: column centres, walls, beam ends, slab edges. Each segment is a span.
- **Splitter:** a polyline that limits the tributaries of lines of the same direction.
- **Tributary:** the slab area a support line designs.
- **Design section:** a cut perpendicular to a span, clipped by the tributary.

Commands (Design ▸ Strip Modelling).

Button	Use
Create X / Y Support Line	click the vertices; C or End finishes and keeps the command, Enter or double-click finishes, Backspace removes the last point, Esc exits
X / Y-Direction Splitter	same keys
Connect Support Lines to Columns and Walls	moves vertices inside a column (5 mm) to its centre, or within half a wall thickness + 5 mm onto the wall centreline (selected lines, or all)
Extend Support Lines to Slab Boundaries	moves end vertices along the end segment to the slab edge, if within 2000 mm
Support Line Tributary Region	select a line, then click the corners of a manual tributary
Parallel Support Lines	asks a spacing, then you draw the first line and click the side
Auto Strips... (large)	makes X and Y support lines at a spacing, with design sections (below)
Delete Support Lines	deletes the selected support lines (and selected splitters) of the shown level, one undo step, "Deleted N support lines."; with none selected it asks "Delete all N support lines of level X?"
Beam Support Lines	on the **Beam ▸ Design** group: adds a support line (criteria Beam) along each beam that has none, then generates the tributaries and sections of the level; asks first if the level has manual tributaries

Radial Support Lines, Delete Design Strips (it removes the tributaries and sections and keeps lines and splitters), Support Line Defaults... and the Punching Table have no ribbon button in this build. The options that new lines take (the table below) can be set in the **Design Section Options** of each line's Properties. A support line has no circle at its first and last vertex (bends keep theirs). A crossing window (right to left) selects the support lines it touches.

Auto Strips... (**Design ▸ Strip Modelling ▸ Auto Strips...**) works on an OpenPT model. It asks the strip spacing in a small window "Auto Strips" (mm, 100 to 20000; it starts from your last answer, 2000 the first time). If support lines exist, it asks Yes (replace) / No (keep and add) / Cancel. For a model of the

older SAFE kind the spacing comes from the **Strips (mm)** box and the command asks for two clicks that define a rectangle ("Auto strips: click the first corner of the rectangle (Esc cancels)"). It makes X and Y support lines with their design sections as one undo step.

Section Design group (Design ▸ Section Design). Four large buttons, in this order: **Generate Sections** (tributaries and sections of every line of the level; replaces manual tributaries), **Use Existing** (Generate Sections Existing: sections only, tributaries kept), **Design the Sections** and **Section Check...** (the calculator of the chapter "Section Check"). The strip design table is exported with **File ▸ Export ▸ Export Design Table (CSV)...**. All strip data are kept in the model's strips sidecar; the INP is not changed. A model made from an ADAPT .adm brings its support lines, splitters, tributaries and design sections with the import.

Strip Layers group (Design ▸ Strip Layers). Display Design Sections is a large toggle ("Show Design Sections"); the small toggles **Show Support Lines**, **Show Splitters** and **Show Tributaries** each have an all, an X-direction and a Y-direction choice. Starting a **Create X / Y Support Line** or splitter tool switches the matching toggle on (an Off becomes that direction, the other direction becomes All; nothing is ever switched off); parallel support lines, Auto Strips and Beam Support Lines also switch the lines layer on for the directions they create.

Strips while the model is locked. Support lines, splitters, tributaries and design sections can be created and edited while the model is locked. OpenPT never asks to delete the results for that; the design becomes out of date and the status bar says **Design out of date**. Geometry, loads and supports are still refused while locked.

Support line options (Support Line Defaults..., or a line's Properties ▸ Design Section Options):

Option	Default
Criteria	Two-way slab (also One-way slab, Beam)
Division criteria	Max spacing (or Max number of sections)
Max spacing	600 mm
Max number of sections	12
Min number of sections	3
Distance from column/wall face	50 mm
Tributary width limit	None (or Fixed per side / Max per side, 1000 mm)
Max construction line spacing	600 mm

Effective flange. For criteria **Beam**, the support line dialog has a **Design Width Limit** "Effective flange" with flags to exclude the tendons and the rebar outside the width. The effective width per section follows ACI (8 h_f, half the clear distance, l_n/8), EC2 and TCVN. It is off by default and ignored, with a note, for the other criteria.

A tendon on the boundary of two design sections counts half of its area in each (strength design, service stresses, cracked deflection, calculated rebar, the design-section panel). A cut end at the slab outline or at an opening is not shared and keeps the full area.

Tributary rule. Each span is divided into at least 3 stations (at the construction line spacing). From each station rays go left and right; the extent on each side is the nearest of: the slab edge (adjoining regions

are not edges), half the distance to the neighbouring line of the same direction, a splitter, or the width limit. The polygon joins the extents.

Section placement. Sections start at the column face + 50 mm (or 100 mm in from a free end). The span is split where the cut starts or stops crossing a column or wall, with a section there. Spacing `s = S / max(⌈S/600⌉, 3)`; each piece gets an odd number of sections so there is always a middle section. Section IDs are `SECTION_ID_<line><span 2 digits><index 3 digits>`, for example `SECTION_ID_102000` = line 1, span 2, first section. A warning lists lines with fewer than two sections.

Covers used in design (from **Tendons & Rebar ▸ Rebar Criteria ▸ Rebar Cover / Size**: minimum cover top and bottom, default 25 mm; preferred bar size top Ø14, bottom Ø12):

Line	Top cover to bar centroid	Bottom
X line (outer layer)	c_top + φ_top/2	c_bot + φ_bot/2
Y line (inner layer)	c_top + φ_top + φ_top/2	c_bot + φ_bot + φ_bot/2

A support line may carry its own covers and bar sizes, which then take precedence. Bar areas use π = 3.14 as ADAPT does (Ø14 = 153.86 mm², Ø12 = 113.04 mm²).

10.2 Strip design (Design the Sections)

Design ▸ Section Design ▸ Design the Sections designs every design section for every design combination and shows the required rebar. It needs a current mesh ("Run Mesh Generation first."), design sections ("No design sections yet. Generate them now?" with the link **Generate Sections**) and at least one design combination. It solves (if needed), computes the section actions, designs, lays out whole bars, reports "Designed N sections (M warnings)." and opens the Results View on Design Sections ▸ Reinforcement ▸ Calculated (required). It runs in the background and is applied only if the strips and the document are unchanged; it can be cancelled. It also runs the one-way shear check, the punching check and the TCVN deflection where the code has them. A model, mesh or strip edit makes the design out of date.

Per combination, per face: each check raises the requirement `Req = max(Req, check)`. The reported service and initial steel is `max(0, Req − base rebar on that face)`. The section requirement is the maximum over all combinations.

INITIAL (transfer). When a face is in tension beyond the initial allowable tension: `As = T / (0.5·fy')`, with T the tensile force of the stress block and `fy' = min(fy, 413.28 MPa)`.

EC2 service (Frequent and Quasi-Permanent).

- *Transfer tension limit*: 0.3·f_ci^(2/3) (EN 1992-1-1 f_ct) in place of ACI's 0.25·√f_ci.
- *Minimum rebar* (EN 1992-1-1 §9.2.1.1): RC sections `max(0.26·fctm/fyk·bt·d, 0.0013·bt·d)`, with bt the mean width of the tension zone between the centroid and the tension face (for T beams the flange counts with at most L/4), fctm = 0.3·fck^(2/3). PT sections with permissible stresses: `As = max(0.0013·b_min·d, 0.26·fctm/fy·d·A_ct/c_t) − Σ(Ap·cos α)·fpk/fy`, less the base rebar of the tension face; A_ct is the tension block of the combination.
- *Crack control* (§7.3.2): `As = kc·k·fct,eff·Act/fy` with fct,eff = max(3.0, allowable), k = 1 (h ≤ 300) to 0.65 (h ≥ 800), `kc = 0.4·(1 − σcp/(1.5·fct,eff))` and σcp = P/A of the section's tendons. In

PT sections it applies when the tension exceeds the allowable; in RC sections (no tendons) whenever the face is in tension.

- *Crack width* (§7.3.4): a cracked-section analysis gives the steel stress, then `w = s_r,max·(ε_sm − ε_cm)` with `ε_sm − ε_cm = max(0.6·σs/Es, (σs − 0.4·fct,eff/ρ·(1 + αe·ρ))/Es)` and `s_r,max = 1.3(h − x)` or `3.4c + 0.425·0.8·0.5·ϕ/ρ` for close spacing; As is iterated until w is just below the allowable width. Frequent applies to bonded PT and RC; Quasi-Permanent to unbonded PT and RC.

ACI 318-14 service (two-way): top steel over supports `0.00075·A_cf` (A_cf from the adjacent span lengths, corrected from the earlier edition, roughly double the old value on some strips) less base steel and bonded tendons nearby; mid-span steel `T/(0.5·fy)` when tension exceeds 0.166·√f'c. One-way and beam service rules are implemented but not verified.

STRENGTH (EC2 and ACI). A layered strain-compatibility design:

- Concrete: EC2 bilinear up to fcd = fck/γc (no αcc), εcu = 0.003; ACI parabola to f'c, εcu = 0.003 (no Whitney block).
- Rebar: elastic–plastic (EC2: fyd = fyk/γs).
- Bonded tendons: strain compatibility with their prestrain (EC2 fpd = 0.9·fpk/γp); force multiplied by cos α of the tendon slope.
- Unbonded tendons: fixed stress `min(σpe + 200, fpy)` MPa.
- The neutral axis is searched until the section capacity matches the target moment within 1 % (ACI target = M/φ). Axial tension adds steel; axial compression is ignored.
- EC2 sections beyond the x/d limit are designed with compression steel and the message "compression steel required (x/d limit of the code reached)". For the other codes, if the moment exceeds what the section can carry even with compression steel at the starting strain plane, the section is reported **"Could not design the section, Check the input"** with no strength steel, as ADAPT does. A search that does not converge gives the warning "did not converge"; such sections are drawn pink.

Rules added by design code.

- *One-way shear* for every code (TCVN 8.1.3.3, EC2 6.2.2, ACI 22.5.5.1; with ACI 318-19 the size effect λs): a **Shear** column in the strip design table and in the CSV; the report headers name the code and the checks run. New output files `shear.csv` and `<code>_deflection.csv`.
- *Minimum rebar of RC strips (no tendons)*: the strength check `Minimum_Rebar_RC` for ACI (ρ·A_g, Table 24.4.3.2; 0.0018 for ACI 318-19) and EC2 (max(0.26·f_ctm/f_yk, 0.0013)·b·d, 9.2.1.1), on the tension face, net of the base rebar ("minimum rebar (...) governs the top/bottom face"). A slab section with no tendon does not get the prestressed-slab rules of ACI 8.6.2.3. The criteria switch `STRENGTH / Minimum_Rebar_RC` turns the check off.
- *ACI*: for two-way PT slabs a check of the minimum average precompression 0.9 MPa (ACI 318-14 8.6.2.1, check `Min_Precompression`, with a message on the first service combination); the cracking-moment strength check 1.2 M_cr for beams and two-way slabs with bonded tendons ("1.2 Mcr (ACI 8.6.2.2): ..."); PT beams: minimum bonded steel 0.004 A_ct on each tension face (unbonded) and 1.2 M_cr (bonded), f_ps limited to f_se + 420 MPa. EC2: 1.15 M_cr for beams with unbonded tendons (9.2.1.1(4)).
- *TCVN and SP 63*: PT strips and sections are designed (bonded tendons at R_sp, unbonded at 0.9 σ_sp with no increase; tendons in the compression half are left out). The slab strip one-way shear between d and 3 h0 from a support uses Q_b = 1.5·R_bt·b·$h0^2$/a. A transfer check `TCVN_Transfer`

("R_bp ... MPa") when there is an Initial combination; a warning for the friction coefficient below the standard table value (SP 63) and, per tendon, when the jacking stress exceeds 0.8 R_s,n (SP 63; nothing is clipped).
- *Unbonded tendons* in strip design use the unbonded rules. *Transfer*: sections over a wall say so in the design table; the 0.9/0.7 transfer factors use external loads only; one P(1)/P(long-term) ratio is used for the model (single-level and multi-level).
- *Plate shears* (V13, V23 and the strip section shear) are recovered from a smoothed moment field, with exact treatment near point and line loads.

Provided rebar. Per line and face, each run of consecutive sections with demand gets `n = [max demand / A_bar]` bars at every section of the run (default Ø14 top, Ø12 bottom). The provided steel is used by Cracked Deflection.

Outputs. Results View ▸ Design Sections ▸ Reinforcement: **Calculated (required)** (envelope or one combination), **Calculated (provided)** (with bar count, e.g. "1234 (8Ø14)"), **Base**. A section's Properties dialog shows per combination: criterion, stresses top/bottom/cg, As top/bottom, design case, crack width and messages. **Export Design Table (CSV)...** writes one row per section and combination, then the envelope and the provided steel.

10.3 Base rebar (Tendons & Rebar tab)

Base rebar is the reinforcement the engineer places in the model; design counts it in every section it crosses.

Group of the Tendons & Rebar tab	Buttons
Rebar Criteria	**Rebar Cover / Size** – minimum cover top/bottom (default 25 mm), preferred bar size top/bottom (Ø14 / Ø12)
Base Rebar	**Mesh**, **Mesh Wizard**, **Banded**, **Distributed**
Generate	**Generate Rebar**, **Rebar Options...**
Rebar Visibility	**Rebar Display Manager**, **Show Rebar**

On a model made from a SAFE file the Base Rebar commands work too; the base rebar is saved in the strips sidecar (key `baseRebar`) and in the `.optp`. Design shows the **required**, **base** and **additional** steel (`AsBase`, `AsAdd`) per face, and **Generate Rebar** draws only the additional steel.

- **Banded:** "Mark the first end of the rebar:", "Mark the second end of the rebar:". Default: bottom face, 1 bar.
- **Distributed:** two clicks for the representative bar, two for the range (extent); the range is kept perpendicular to the bar. At least 2 bars; spacing = range/(n − 1).
- **Mesh:** click the corners; C, Enter, End or double-click closes. The mesh must lie over a slab region ("The mesh must lie over a slab region."). Defaults: top, As1 = As2 = 200 mm²/m, CGS1 25 / CGS2 35 mm, angle 0.
- **Mesh Wizard** ("Create Mesh Reinforcement Automatically"): one mesh over each selected slab region, or over **All slab regions of this level**.
- Bar library: 6–40 mm, area $3.14 \cdot d^2/4$. All commands repeat until Esc.

Properties: **Rebar** (Base/Calculated, Top/Bottom, bar size, Number of bars, Spacing, cover, Distributed/Grouped, extents, bar description options) and **Mesh Reinforcement** (Top/Bottom, CGS1, CGS2, Area or Bar size with spacing, angle to mesh axis 1-1).

How base rebar enters a design section. A mesh crossed over a length L gives `A = (A1·c1² + A2·c2²)·L`, where c_k is the cosine between the cut normal and the bar direction k; its height is the area-weighted CGS from the face of its slab region. A bar that crosses the cut (5 mm tolerance) counts fully: `A = n·A_bar·cos²ϕ`. There is no development-length reduction. Base rebar is used by strip design and cracked deflection; calculated bars (source "Calculated") are not counted as base rebar.

10.4 Generate Rebar and Rebar Options

Tendons & Rebar ▸ Generate ▸ Generate Rebar turns the required steel of the design sections into real bars, stored in the model as calculated rebar. It generates at once with the last choices of the session, without a dialog, as one undo step, switches the rebar display on and shows the bars in the Plan and in 3D. **Rebar Options...** (the same group) opens the dialog "Generate Rebar Drawing Options" and generates when you press OK; the dialog opens with the choices of the last OK on the same document (the choices live for the session only). Messages if something is missing: "No Support Lines exist. You need to create "Design Section(s)" first.", "No solution available. Analyze structure first.", "No rebar drawing is created. Design the design sections first." The command refuses while the results were made with a load case skipped (see "Analysis Options") and in Multi-Level mode.

Dialog "Generate Rebar Drawing Options":

Option	Default
Load Combination	Envelope (also Envelope Service, Envelope Strength, each design combination)
Bar Length Selection	Calculated Lengths (or Library Lengths: 2000, 3000, 4000, 5000, 6000 mm)
Bar Orientation	Along support lines (or Angle from global x-axis: X-direction 0°, Y-direction 90°)

For a model of the older SAFE kind the dialog offers the load or combination (**Strength + minimum steel** by default, or **Strength only**), the bar size top and bottom (default the design bar, 12 mm) and the strips used (drawn support lines or uniform strips), and reads the cover from the slab property.

Generation per line and face:

1. Walk the sections in order; a **run** collects consecutive sections with demand, joined when the gap is shorter than max(2 × extension, local section spacing). A run covers one section before and after its demand.
2. Count `nb = ⌈max demand of the run / A_bar⌉`; all nb bars run the full length (no stagger).
3. Bar length: the run length + 2 × 300 mm extension, anchored about the one-third point, rounded up to 50 mm (or to the library length). Over a column the bar is placed across the column (PT models).
4. A bar end outside the slab slides back onto the edge.
5. Banded or distributed: in PT models, top bars, beam-line bars and bars at columns are banded; bottom bars are distributed unless the tendons crossing the section are bundled. In RC models all bars are distributed (spacing rounded down to 25 mm). The status line warns "RC mode: bar layout is approximate."

The previous calculated bars of the level are replaced. Bars that should sit in a thicker part of the strip are drawn blue with a message. Calculated bars are drawn **without** the two end circles that base bars keep, and they can be edited. When the design changes, the info line says the calculated rebar may be out of date and tells you to generate it again. With a single combination selected the note "For the selected load combination, rebar changes will not be reflected in the envelope." is shown.

Punching. Generate Rebar also runs the punching check on the existing solution (no new analysis); the status line says "Calculated rebar: ...; punching: 18 columns, max ratio 0.93, 4 with reinforcement". If punching cannot run, the rebar is still made and the status adds "Punching not checked: <reason>". Punching reinforcement is always calculated when a column fails without it, for every code. Showing it is the option **Display punching reinforcement** of the **Rebar Display Manager** (below "Display base rebar"); **Show Rebar** off hides it too.

Status lines and time. On a large model Generate Rebar computes off the user-interface thread with the status "Generating rebar... (Esc to cancel)", then "Writing N bars into the model...", "Checking punching shear...", and the final text "Calculated Rebar Plan (<load>): N bars (T top, B bottom)." After 60 seconds it says "Still generating rebar (N s)... Esc to cancel"; nothing is stopped automatically. A second Generate Rebar on an unchanged design and unchanged options ends in about 50 ms with the same text. When it takes 3 seconds or more, a line with the phase times goes to `timing.log` next to `error.log`.

Fed with the same required steel, the generator reproduces ADAPT's bar plan (counts, lengths, positions) on the tested projects; differences in steel mass come from differences in the required steel, not from the layout.

10.5 Cracked deflection

Analyse ▸ Run ▸ Cracked Deflection estimates the cracked sustained and long-term deflection. It needs a mesh, support lines and design sections ("Define support lines and generate design sections first"). Beams are covered too (EC2 7.4.3 and TCVN 5574:2018 8.2.3 methods; EC2 beams default to Branson × (1 + φ)).

Steps:

1. Solve every load case; take the design combinations (and the defaults when missing).
2. Compute the section actions and run the strip design; take the **provided** rebar.
3. For each design section under the **sustained** combination (the Cracked Deflection combination; default 1.0 Selfweight + 1.0 Dead + 0.3 Live + 1.0 Prestressing): - modulus of rupture: EC2 `fr = fctm·max(1.6 − h/1000, 1.0)`, ACI `fr = 0.62·√f'c`; - cracking moments `Mcr+ = (fr + P/A)·Ig/c_bot`, `Mcr− = (fr + P/A)·Ig/c_top` (P from the section's tendons); - cracked inertia Icr from a cracked-section analysis with tendons, base rebar and provided rebar (tension stiffening limit: ACI 0.5·√f'c, EC2 fctm); - **Branson:** `Ie = (Mcr/Ma)³·Ig + (1 − (Mcr/Ma)³)·Icr ≤ Ig`; ratio `R = Ie/Ig`.
4. Every slab shell and beam frame takes the ratio R of the nearest section of the strip whose tributary contains it, for each of the two directions.
5. The floor is solved once with the reduced stiffness → **Cracked sustained**.
6. **Long term** = (1 + φ) × cracked sustained, with φ from the Long-Term combination (factor on the sustained combination − 1; default 2.0, so 3.0×).

Outputs: the Load drop-down gains **Cracked sustained**, **Cracked long-term** and **Sustained (uncracked)**. The status line gives the maxima, the cracked/uncracked ratio and the number of cracked sections. **Crack ratio** colours every shell by min(R1, R2) (0 = fully cracked, 1 = uncracked). **File ▸ Export ▸ Export Cracked Table...** writes per section Ma, Mcr, Ig, Icr, Ie, R and the required and provided rebar. Editing the strips or the rebar cover/size marks the result out of date.

Agreement. On a tested project the cracked sustained and long-term deflections were within about 3 % of ADAPT's.

10.6 Punching shear

Analyse ▸ Punching has **Shear Design Options**, **Shear Check** and, on a model of the older SAFE kind, **Punching bars...** (the Punching Table has no ribbon button). The check needs a mesh of the shown level. **The punching code is the document design code** (**Design ▸ Criteria ▸ Design Code...**): TCVN 5574:2018 and SP 63 use the check of the chapter "TCVN 5574:2018 design"; EC2, ACI 318-14 and ACI 318-19 use the rules below. A punching code that an older model had set separately is kept as "Punching code (legacy)" with a remove button. Results: Results View ▸ Punching Shear (**Stress check**: green OK, orange Reinforce, red Exceeds Code, grey NA; **Stress ratio**; **Condition**: Interior / Edge / Corner; **Reinforcement**; **Reinforcement layout**: the legs, the outer contour and the label). The status line gives "Punching: N checked, F fail, K not checked (max ratio R at <label>)". The loads are the column forces of the single-level run **including the force of the member above**. The messages that send you to the check read "Run Analyse ▸ Punching ▸ Shear Check first." and "Run Shear Check (or Calculated Rebar Plan) to get a punching result."; after a change they read "Punching bars changed: run Shear Check (or Calculated Rebar Plan) to apply them." and "Shear Design options changed: run Shear Check again to apply them.", and if the model changed during the check, "The model changed during the check: run Shear Check again."

Shear Design Options (kept per file in OpenPT's settings, not in the INP):

Option	Default
Precompression σcp	Default (0.862 MPa with tendons, else 0), or a Value (MPa)
Flexural reinforcement ratio ρl	0
Top cover + bar (d = h − this)	From Criteria ▸ Rebar Cover/Size (cover + bar, else 25 mm), or a Value
Reinforcement	Studs (or Stirrups), bar Ø10, N_Ds = N_Dr = 2, α 90°, fy of the rebar material (500 if unknown)
Combined stress check (rr and ss)	off

Studs and TCVN stirrups are as described before; for EC2 and ACI stirrups the **Reinforcement layout** draws **every stirrup row** (legs as dots, evenly along the perimeter at the row's distance from the column face), and the label and the punching report table read "<n> rows Ø<d> stirrups, 1st row <a> mm from face, rows @ <s> mm, <N> legs @ <= <t> mm" (the spacing is a range when rows differ; with one row there is no "rows @" part).

Which supports. Every column below the slab, planted columns above with nothing below, and short walls (length/thickness ≤ 4). **Not checked:** a column touched by a beam deeper than the slab or by a wall, a column the slab edge cuts into, a support with no slab under it. A beam is deeper than the slab when its bottom lies more than 1 mm below the slab soffit at the column. Offsets are measured down

from the level's reference plane: the beam bottom is the beam offset at the column plus the beam depth, and the soffit is the region offset plus the punching thickness h (the drop cap's bottom when the column sits in a cap). For a beam and slab flush with the top this means depth > h. A column with only shallow beams (depth ≤ h, equal depth included) is checked exactly like a flat-slab column: the same perimeter, h and reinforcement design; the shallow beam does not cut the perimeter. ADAPT skips every column a beam touches, so these columns have no ADAPT counterpart. The slab thickness is that of the smallest region at the support, reduced when a drop panel ends close to the column. d = h − (top cover + bar). The loads are the column forces at the slab node (lower plus upper column) for every Strength combination, with moments resolved on the column axes.

Classification. A side is a free edge when the face distance to the slab edge (or opening) is ≤ 4h. Free edges on both axes = corner; on one = edge; none = interior.

EC2 (EN 1992-1-1 §6.4).

- β = 1.15 interior, 1.40 edge, 1.50 corner (simplified β; moments are not used separately).
- Face: `v = β·VEd/(u0·d) ≤ vRd,max = 0.4·ν·fcd` (EN 1992-1-1 6.4.5(3)), ν = 0.6(1 − fck/250).
- Basic perimeter at 2d (rounded corners), then layers every 0.75d: `vRd,c = max(0.18/γc·k·(100·ρl·fck)^(1/3), 0.035·k^1.5·√fck) + 0.1·σcp`, k = min(1 + √(200/d), 2).
- Reinforcement where φ·vRd,c < v ≤ the upper limit: `As = (v − 0.75·vRd,c)·u·s/(1.5·fywd,ef·sin α)`, fywd,ef = min(250 + 0.25d, fy/γs); studs per rail and spacing, or stirrups.
- Combined option: β' = 1 + 2(β − 1) for a biaxial block.

ACI 318-14 (and 318-08). Layers at d/2, d, 3d/2, ...; rectangular critical sections open on free sides; `v = Vu/(b0·d) + γv·Mu·c/Jc` per axis (axes not summed), `γv = 1 − 1/(1 + ⅔√(b1/b2))`. RC: `vc = 0.083·λ·min(4, 2 + 4/βc, 2 + αs·d/b0)·√f'c`; PT interior with σcp > 0.861 MPa: `vc = 0.083·λ·min(3.5, αs·d/b0 + 1.5)·√f'c + 0.3·σcp`. Studs with vcs and vmax limits, or stirrups.

Circular columns are checked as well (ACI $b_o = \pi(D + d)$; EC2 $u_0 = \pi D$ and $u_1 = \pi(D + 4d)$; TCVN a circle at h0/2), cut by the real slab edge for edge and corner columns, and with drop caps. The stiffness of a round column is exact ($\pi D^4/64$).

Punching Table (command without a ribbon button): one row per support (Label, Kind, Class, Status, Max ratio, combination, layer, axis, Governing As, Warnings); expand a row for its combinations and layers; **Export CSV...**.

Limitations. Drop caps are not modelled as capitals; columns with edge beams are not checked.

10.7 Beams without a slab (the Beam tab)

The **Beam** tab (the last tab) is for beams and frames with no slab under them, and for beam work on any level. Its groups are: **Model** (**Column**, **Beam**, **Support**), **Loads** (**Add Point Load**, **Add Line Load**, **Line Load Wizard**, **Load Cases**, **Load Combinations**), **Tendon** (**Add Tendon**, **Trim/Extend Tendon**), **Analysis** (**Mesh size (mm)**, **Mesh and Analyse**), **Results** and **Design** (**Beam Support Lines**, **Generate Beam Rebar**, **Beam Rebar Options...**, **Show Beam Rebar**). **Section Check...** is no longer on the Beam tab: it is in **Design ▸ Section Design**.

A level may hold beams only. Point supports sit on the beams (see "Supports"); the mesh is a frame-only mesh. **Mesh size (mm)** (caption on its own line above the box, 100 to 20000) is the largest element

length of **Mesh and Analyse**, which meshes with the box value when the mesh is missing or stale and then analyses. **Run Analysis** (F9) itself is unchanged. Results: **Beam moment**, **Beam shear**, **Beam axial force** and a **Beam deflection** diagram. On a level with no slab the shear diagrams, design stations and tables read the **statics of the member**: a uniform load gives a straight line with V_max = qL/2, with jumps at point loads and the true end values at supports; moments, deflections and reactions are unchanged and slab models are identical. An inclined beam (slope of 1 % or more) on a level with no slab is read normal to its axis (V) and along it (N); beams under a slab are read as before and named in the design warnings.

10.8 Beam design (Generate Beam Rebar)

Beam ▸ Design ▸ Generate Beam Rebar designs reinforced and post-tensioned beams for flexure and shear (stirrups) under TCVN 5574:2018, SP 63, EC2 and ACI 318-14/19, both for the beams of a floor and for beam-only models. It runs with the saved options and no dialog. **Beam Rebar Options...** opens the dialog "Beam Rebar Options" and then runs. **Show Beam Rebar** shows or hides the required steel, the bars and the stirrups on the plans; it is off and disabled on a model with no beams.

Options:

Option	Default
Design code	read-only (with a **Design Code...** button)
Longitudinal diameters	16 18 20 22 25 28 32
Stirrup diameter	10
Legs	Auto: 2 for b < 400 mm, else 4
Clear cover	25 mm (TCVN, EC2), 40 mm (ACI)
Station spacing	min(500 mm, L/10)
Crack width	on/off, with the service combinations

Results. A window "Beam Design" with the tabs **Beam Design** (per span) and **Beam Sections** (per station), exportable as CSV (`beam_design.csv`, `beam_sections.csv`; the column `d_mm` is the effective depth of the governing face). The beams on the plan turn green (OK), orange (warning) or red (FAIL) with bar labels; the status line says "Beam design: 24 spans, 22 OK, 2 FAIL (...)". Messages: "The model has no beams.", "Run the analysis first.", "There is no strength combination to design with." On the command line, `--beam-design` makes the same run.

What it covers.

- Extra design stations at point loads and at the start and end of partial loads (two stations at one place, before and after the jump). Envelope combinations are designed through their members.
- **T sections**: flange width and thickness are stored in `<model>.beamdesign.json`; flexure, capacity and crack width use the flange on levels with no slab; prestressed beams do not use the flange. Beams under a slab level are not covered. For criteria Beam of a support line there is also the "Effective flange" width limit.
- TCVN stirrup design follows the textbook rule (the curve Q_b(c), a scan of the inclined crack and the maximum spacing). TCVN and SP 63 beam deflection uses the T-section flange and the humidity of the criteria, and, optionally, the bars you give in `beamdesign.json` (`"bars"`: `B1=bottom 769@27; top 0@30`; a malformed entry stops the run with a message).

- ACI PT beams: minimum bonded steel and f_ps (see "Strip design"). EC2: compression steel when x/d reaches the limit.
- A beam on a model made from a SAFE file with no slab over it is designed with the shear and moment of the statics (the support shear equals the reaction); a beam partly under a slab keeps the reading per element.
- Messages from the engine (for example new TCVN warnings) are in English only.

10.9 Section Check

Design ▸ Section Design ▸ Section Check... (a large button after **Design the Sections**; it shows for INP and F2K documents, and with no document open) opens a window that checks **one section with forces you type**, with no analysis needed. It is a standalone design and verification calculator powered by OpenPT's design engine: the **kinds** of check (flexural design and capacity, one-way shear and stirrups, crack width, curvature, punching (interior and edge), simplified deflection, rectangular, I, ring, circular and biaxial column checks, torsion, local bearing, oblique sections, bar cut-off and anchorage, a trapezoidal rib, and others), with inputs named with their units and a results table with the manual's formula references. **Copy table** copies the table (tab separated; no file is written). TCVN 5574:2018 is offered only for the kinds whose formula is in TCVN; the others are SP 63 only.

Design code. The **Design code** box of the window lists five entries: SP 63.13330.2018, TCVN 5574:2018, EC2, ACI 2014/IBC 2015 (ACI 318-14) and ACI 2019/IBC 2021 (ACI 318-19). There are two families of kinds. SP 63 and TCVN show the 33 kinds above, in 8 groups. **EC2 and the two ACI codes show 6 kinds**, valid for all three codes:

Group	Kind	Inputs (mm, MPa, kN, kN·m)	Results
Bending	Slab: required reinforcement	b, h, d, fc, fy, M	As, AsMin, AsDesign
Bending	Beam: required reinforcement	b, h, d, fc, fy, M, bf and hf (flange; 0 = none)	As, AsMin, AsComp, AsDesign
Shear	Beam: stirrups	b, h, d, As, fc, fy, fys, V	Av/s, Av/s min, s max, φVc, φVn max, VRd,c, cot θ
Punching	Punching: interior column	c1, c2, d, fc, V, M2, M3, rho1 (EC2 only), sigmaCp, combined (biaxial moments, ACI)	b0, u1, vu, capacity, ratio (Pass or Fail)
Post-tensioning	PT section: required reinforcement	b, h, d, dp, fc, fy, fpu, fpy, fse, Aps, M, bonded, span/depth	As, AsStrength, AsMin, tendon capacity
Post-tensioning	PT section: service stresses	b, h, P, e, M	top and bottom stress

The engine returns either the ACI rows or the EC2 rows of the stirrup and punching kinds, so the table lists only the rows that apply to the chosen code. For these kinds the Concrete, Rebar, Stirrups and Humidity boxes, the strength overrides and the "T = also valid under TCVN" legend are hidden: fc and fy are inputs of the form, and the code table values are used (the EC2 partial factors cannot be overridden in this window). Switching between EC2, ACI 2014 and ACI 2019 keeps the kind and the values you typed; switching between the two families lists the kinds again and returns to the last kind of that family. The **Example** box offers the cases of the verification set for each kind (T1 to T8, from the SAFE 22 verification manual); loading one sets its code, fills the form, shows the manual's value in the Manual column and calculates. A case with the factors gc, gs or acc loads without them (they are not inputs), so

its result may differ from the manual column. The window opens on EC2, ACI 2014 or ACI 2019 when the open model's design code is one of them; for the other ACI editions (2005, 2008, 2011) it opens on SP 63, and the model's concrete and steel are not copied into the EC2 or ACI form. **Copy table** gives the design code as its materials line. The command line `--section-check <cases.json>` also accepts `"code": "ACI14"` or `"EC2"` (slab flexure, beam flexure, beam shear, punching, PT flexure, PT stress).

10.10 Time-Dependent Analysis (ACI 209)

Analyse ▸ Run ▸ Time-Dependent Analysis... evaluates stage-by-stage creep, shrinkage, prestress relaxation and long-term deflection of a PT beam or strip. It is available for OpenPT models with support lines (not for the older SAFE kind), needs the licence feature "analysis", and opens a window like Section Check with:

- the **history and material inputs** (defaults: stressing at 7 days, sustained load at 28 days, end time 20 years; ACI 209R-92 creep and shrinkage; relaxation after Magura);
- the frame built from the support line;
- the **results**: the tendon stress along the tendon at the output times, the PT, hyperstatic and time-effect moments, the deflection over time and the fibre stresses, with per-stage tables, plots and an **Export CSV**, in English or Vietnamese.

The limits are stated in the report: every vertex is a pin or roller (no column rotational springs), rebar layers are not offered in the dialog, and unbonded tendons use an average strain.

10.11 Calculation Report (.docx)

File ▸ Export ▸ Calculation Report (the same command as the **Full Report** button of the Compiled Reports group on the **Report** tab; in Vietnamese "Phụ lục tính toán" / "Thuyết minh đầy đủ") writes a Word `.docx` calculation report of the slab, in the program's language (English or Vietnamese), without starting Word: the plan, the tendons, the loads, the deflection, the rebar, the design strips, the punching and the one-way shear, with pictures. The default name is `PLTT_san_<model>.docx`; the info line says "Calculation report written: ...". It works for an OpenPT model, with one chapter per level of the results (with the level's name, and "(with openings)" when the level has openings).

It **refuses**, writing nothing, when the analysis or the strip design is missing or out of date ("The analysis or the strip design is missing or out of date. Run Analyse (F9) and the strip design, then write the report."). Punching and cracked deflection that are missing or stale do not block the report: that section holds one sentence that says to run them, and the info line adds "sections not run". The report sentences follow the rules of all messages (they say what happened and what to do); the tables keep technical notation. Every model-sized label in report pictures is at least 20 px high (about 2 mm on the 16 cm figure) and the figure title is 28 px.

10.12 Reports and the Report Compiler

The **Report** tab holds:

- **Print**;
- **Compiled Reports**: **Open Report Compiler**, **Take Report Screenshot**, **Full Report** (the calculation report above);

- **Default Report** (drop-down menus): General (Report/Project Title Settings, Report Cover, Material and Design Criteria, Quantity and Cost), Structural (slab, opening, beam, column, drop and wall reports, Releases and Restraints, Plan Geometry), Loads (Load Cases and Combinations, Applied Loads, Dead, Live, Other), Tendon (Tendon Report, Elongation, Totals, Plan), Rebar (List, Totals, Design Section Rebar, Punching Shear Reinforcement, Plan), Punching Shear (Stress Check Parameters and Results, Reinforcement), Column and Wall (reactions);
- **Analysis Reports**: Design Strips (forces, rebar, capacities, dimensions, geometry, support lines, strips, checks X and Y), Analysis Data (Equilibrium Check, Solution Statistics, Nodes, Elements, Mesh Plan) and Analysis Reactions (point and line support reactions);
- **Export ▸ Export to DWG**;
- **ETABS** (below).

The **Report Compiler** ("Report Generation Manager") has a tree with Cover, Content List, Tabular, Graphical, Screenshots and Imported (**Import Image...**, **Import Text File...**); **Move Up**, **Move Down**, **Remove**, **Remove All**; **Generate RTF Report** (`Report\CombinedReport.rtf`) and **Generate PDF Graphical Report** (`CombinedReport.pdf`). The text of these reports is English. Items that are not available for the open document are greyed, with the reason.

10.13 Sending a floor to ETABS

Report ▸ ETABS has three large buttons: **Send to ETABS...**, **Get ETABS Forces...** and **Import ETABS Forces from File...**. They work on an OpenPT model with a support line. The plan offset between ETABS and OpenPT is found from the columns and shown in two boxes of the dialogs. The forces are kept in a `.etabs_forces.json` sidecar that travels inside the `.optp`, and the strip design takes the ETABS moments in place of OpenPT's where it holds them.

11 Verification against ADAPT

The separate **Verification Manual** (30 illustrated cases) and **Verification Manual - full** (188 cases) are in **Help** ▾ ▸ **Documentation...**. This chapter is the summary of the comparison with ADAPT-Builder 2019.

OpenPT has been compared with ADAPT-Builder 2019 on a benchmark set: ten synthetic floors (B01 to B10F, PT and RC, with walls, beams, drops and openings) and six real project floors. Each case is analysed and designed by both programs with the same model and settings, and the results are compared on four measures:

Measure	What is compared
M1	cracked long-term maximum deflection
M2	ratio of cracked to uncracked sustained deflection
M3	calculated rebar: required steel per section face, share of section faces that agree, total steel mass top and bottom
M4	punching shear: classification, stress ratios and status per column

Synthetic floors. All 40 measure cells (10 cases × 4 measures) pass their tolerances. Deflections agree typically within 1–2 %.

Real projects (latest results; ADAPT in brackets):

Project	M1 long-term deflection	Section faces agreeing (M3)	Steel mass top / bottom (kg)
XuanLa	23.25 mm (23.44)	100 %	35.8 / 5.8 (35.8 / 5.8)
H1HH1	54.59 mm (54.49)	100 %	3.0 top (3.0)
B3CC1	36.03 mm (35.96)	93 %	2426 / 898 (2576 / 952)
HongNgoc	29.1 mm (27.8)	99 %	325 / 104 (339 / 100)
EMASI	37.33 mm (37.01)	92 %	8561 / 7898 (8686 / 7904)

What agrees exactly or nearly exactly:

- tendon friction, seating and elongation (median differences well below 1 %);
- the mesh, when the same cell size, pattern and node shift are used;
- the solver on the same model (checked to a 1 % tolerance);
- the section-action integration, given the same element forces;
- the strip designer, given the same section input (99–100 % of rows for initial, service and strength);
- the calculated rebar layout, given the same required steel.

Known differences (what the user should expect):

- Most remaining differences in required steel and steel mass come from small differences in the **analysis input**: the node shift of a few members, where the analysis nodes of beams and columns end up; the prestress loads of tendons that run inside beam strips; and walls under deep beams. They concentrate on a few support lines per project.

- A section whose required steel is close to a bar-count step can change by one bar with a 1–2 % change in moment; small masses (a few kg) are sensitive to this.
- If the ADAPT model contains members not connected to anything (floating beams), ADAPT's own results on the strips crossing them are not reliable, and a comparison there is meaningless.
- One-way and beam service rules for ACI, twisting moments in design, and shear capacity of one-way and beam sections are not compared (not implemented or not verified).
- Punching: the class (interior, edge, corner) and the status agree on the tested columns. The stress ratio follows the column force directly, so any analysis difference at a column (for example a different slab thickness under it) shows in its ratio.

Practical advice. To compare OpenPT with an ADAPT run, use the same cell size, mesh pattern (Sparse/Uniform) and maximum shift distance as the ADAPT run, keep the ADAPT .adm file next to the INP (OpenPT then follows ADAPT's tendon hosting), import the .adm with **File ▸ Import ▸ Import model…** (its support lines come with it), and check the load balance and node-shift messages in the **Check Model** window before comparing results.

Changes since the comparison. Some design rules changed after the tables above were made, to follow the codes or published worked examples more closely: the ACI two-way support-zone minimum rebar, the EC2 transfer tension limit, the ACI minimum average precompression, the cracking-moment checks, the minimum rebar of RC strips, ACI 318-19, the EC2 punching limit at the column face and the plate-shear recovery (see "Strip design" and "Punching shear" in the chapter "Design"). The ADAPT-compatible rules that the comparison used are still available as an internal switch, which is off in the shipped program. The strength capacity of PT sections is 2.4 to 3.7 % conservative against ADAPT because of its pivot rule; this was kept by decision. In the multi-level analysis of the program's own step, the node counts differ from ADAPT's by design (case M01: 916 nodes against 856), because a column whose centre lies on the slab is never moved by the node shift.

12 Working with SAFE models

OpenPT works with floors that ETABS or SAFE exported as a **.F2K** text file, a **$sf** table file or an **.FDB** database (SAFE 12, 16, 20 and 22). There is no separate "SAFE" mode or tab any more: **File ▸ Import ▸ Import model...** converts the file into a normal OpenPT model, and every command of the ribbon works on it. OpenPT never writes into the SAFE file. Everything it adds is kept in small sidecar files and in the `.optp` project that you save.

This chapter covers what the import reads and makes, the sidecar files, how to work with the imported model, the way back to SAFE and ETABS, and the groups of an older kind of document that you can still meet in an old `.optp`. The TCVN design checks are in the chapter "TCVN 5574:2018 design".

12.1 1. Importing a SAFE file

1. **File ▸ Import ▸ Import model...** and pick the file (the filter lists all model files; OpenPT looks at the content, not at the extension). A file given on the command line, dropped on the window or clicked in **Recent files** does the same.
2. OpenPT converts the file with its default converter options. There is **no dialog**. The result is a new **untitled** model of the INP kind, marked as changed, with no link to the source file.
3. The info bar gives the counts, and the import report goes to the **Check Model** window (**Analyse ▸ Check ▸ Check Model**, F5) and to the **Report** tab of the Results View pane. The report is also written to `<name>.import.txt` next to the source file (when the folder can be written). It has three parts: - **What OpenPT read from <file>**: the counts of points, lines, areas, patterns, cases, modes and combinations; - **What was converted**: what became a wall, a column, a beam, a load, a case or a combination; - **What was skipped, and what to do**: records OpenPT does not use, with the line number and a hint.
4. Save the model as an `.optp` (**File ▸ Save As**) to keep it.

What the import makes. The converter options of the import are fixed:

Item	Result
Wall lines (a line named like `W..-W`, or a beam section at least 3000 mm deep)	Walls below and above the slab, of the thickness B of the beam section.
Restrained points	Columns below and above the slab, of the size DIMX × DIMY of the point (or the columns sidecar, or 400 mm).
Other lines	Beams.
Storey heights	3300 mm below and above the slab (LEVEL_2 is the slab level).
Openings	Taken from the openings sidecar when there is one.
Mesh cell	600 mm (change it in **Analyse ▸ Meshing**).
Loads, cases, combinations	All converted (table in section 3).

The imported model is an ordinary OpenPT model. Tendons, rebar, strips, Auto Strips, Generate Rebar, punching, cracked deflection, Multi-Level analysis and the TCVN checks all work on it.

> Imposed displacements of the file (`POINTDISPL`) are read and kept but **not applied** in the analysis (spec 237). The import report lists them. This matters for the `*_ABOVE` patterns that ETABS writes, which carry imposed displacements of about 20 mm.

12.2 2. What OpenPT reads from a SAFE text file

OpenPT reads every record that ETABS or SAFE writes into a floor file:

SAFE file content	What OpenPT does with it
Units (`UNITS N mm`, also kN, kgf, tonf, lb, kip and m, cm, in, ft)	Converted to N and mm on reading.
Titles	Kept for display.
Grids (`GRID`, `GRIDSYSTEM`)	Kept with the file and drawn on the plan as grid lines with bubbles and spacing dimensions.
Beam sections (`BEAMPROP`, rectangular)	Frame sections b × d. Any non-rectangular section type stops the analysis with a clear error.
Slab sections (`SLABPROP`)	Thickness (TII), design thickness and covers (CTI, CBI), E, ν, unit weight. A "column" type slab property is read as SAFE's "Stiff" property.
Point objects (`POINT`) with restraint flags and `DIMX/DIMY`	Joints; each restraint flag fixes that direction. `DIMX/DIMY` (column size) are punching data only. They have no stiffness effect.
Line objects (`LINE` + `BEAM`)	Beams (frame elements) along the line.
Area objects (`AREA` + `SLAB`)	Slab regions. Coordinates may continue on following lines.
Load patterns (`LOAD … TYPE … SELFWEIGHT …`)	Load patterns with their type (DEAD, SUPERDEAD, LIVE, WIND, QUAKE …) and self-weight multiplier.
Point, line and area loads	All six force components. Several records for the same point and pattern add up, as in SAFE.
Imposed displacements (`POINTDISPL`)	Read, kept and exported (all six components); not applied in OpenPT's analysis (spec 237).
Linear static cases (`LINSTAT`)	Load cases = sum of factor × pattern. Cases whose name starts with `~` (for example `~Dummy`) are dropped, as SAFE does.
Modal case, response spectrum functions and cases	Imported modes and response spectrum cases (CQC over the modes, SRSS over the directions).
Combinations (`COMBO`, `COMBOFACTOR`)	Linear-add combinations. A combination that contains a response spectrum case has a **Max** and a **Min** step.

Sign conventions follow SAFE: FZ and UZ in the F2K are positive downward (gravity). OpenPT shows the global Z axis upward.

Any line OpenPT does not understand goes into the **unread records** list of the import report, with its line number. A well-formed ETABS export imports with zero unread records.

12.3 3. Behaviour of the converted model

The conversion reproduces what SAFE does with the same file, except that walls and columns are real members. The base reactions of the converted model agree with SAFE's to better than 1e-6 relative on a real tower floor, and joint displacements and support forces agree closely (p90 of the error below about 1 % for gravity). The shell moments M11/M22 differ by a few percent of the maximum (more at point supports and in very thin slivers of slab), because the plate elements differ. Element-corner shears V13/V23 are not a reliable estimate of the true shear in either program: do not design for shear from them.

The rules that carry over from SAFE:

1. **One plane, no offsets.** The slab mid-plane, the beam axes and the joints lie at z = 0 (SAFE's "Ignore offsets"), so there is no T-beam action.
2. **Beam shear deformation is on** (shear areas 5/6 A) and the **torsion constant is one tenth of the tabulated J**, as SAFE's results show.
3. **Beam self-weight** uses W × b × d × length of the full rectangle, so the part of a beam inside the slab is counted twice, as in SAFE.
4. **Point loads with XDIM/YDIM** act at the joint; the dimensions are punching data only.
5. **A point that is on no element** (for example inside an opening) gives its load to the shell that contains it. If no shell contains it, the load is dropped, as SAFE does; the import report says so.
6. **Drilling rotation** of the slab joints follows the membrane rotation, as in SAFE's shells.

The import report names the choices under "SAFE behaviour, observed".

12.4 4. Sidecar files: openings and columns

ETABS exports usually carry no slab openings and no column sizes: ETABS models a column as a wall line with fixed points, without a size. OpenPT reads both from two optional JSON files placed **next to the SAFE file with the same stem** when you import:

File	Content	Used for
`MyTower_L10.openings.json`	Slab openings (polygons).	Mesh holes, loads, strip sections, punching contours.
`MyTower_L10.columns.json`	Column rectangles and the points that carry each column.	Column size and place, punching, plan and 3D drawing.

The import report says what was loaded, for example "n openings, area m²; n columns".

4.1 Openings file

{ "units": "mm", "openings": [{ "name": "ME001", "polygon": [[x, y], [x, y], [x, y], [x, y]] }] }

```
- Coordinates are the model coordinates of the SAFE file. `units` may be `mm`, `cm` or `m`.
- Polygons have 3 or more vertices, are closed implicitly and may run in either direction.
- An opening removes slab from **every** slab area it overlaps. Overlapping openings are
merged (union).
- Area loads and slab self-weight act only on the remaining slab.
- Beams that cross an opening are kept, with their own self-weight and loads.
- Openings smaller than the mesh cell (for example 150 × 150 mm sleeves) are still cut as
holes, with mesh nodes at their corners.
- The import report gives the slab area before and after, and the opening area actually
removed.


After the import the openings are ordinary OpenPT openings (**Model ▸ Add Structural
Components ▸ Opening** and the model tree), which you can edit.

### 4.2 Columns file


```json
{ "columns": [ {
    "name": "C5ACEB",
    "x0": 43680, "y0": 32350, "x1": 44180, "y1": 33850,
    "bx": 500, "by": 1500, "cx": 43930, "cy": 33100,
    "f2k_points": ["11529", "11530"],
    "beams_framing": []
} ] }
```

Key	Meaning
`name`	Column name.
`x0`, `y0`, `x1`, `y1`	The column rectangle (mm).
`bx`, `by`	Column size.
`cx`, `cy`	Column centroid.
`f2k_points`	The F2K point supports whose reactions make up the column force.
`beams_framing`	The lines of the beams framing into the column. An empty list means a flat-slab column. A beam whose section depth D is not more than the slab thickness h of the punching row (D ≤ h) is shallow and counts as no beam for the filter; a line that is not found or has no section depth counts as deep (a design warning names it once).

Other keys (for example `f2k_wall_lines`, `frac_outside_slab`) are allowed and ignored by the check.

The column force for punching is N = the sum of the vertical reactions of its `f2k_points`. Mx and My are the point moments plus the couples of the vertical reactions about (cx, cy).

Without a columns file, every restrained point becomes a column of the size DIMX × DIMY of the point, or 400 mm. After the import the columns are ordinary OpenPT columns (**Model ▸ Add Structural Components ▸ Column**).
```

4.3 Other files written by OpenPT

File	Holds
`<name>.import.txt`	The import report.
`<model>.openpt.json`	Point supports and springs, imposed displacements, cases made of several patterns, modal and response spectrum cases, cover values (see section 5).
`<model>.combos.json`	Envelope, SRSS, nested and spectrum-based combinations.
`<model>.strips.json`	Drawn support lines, splitters, design sections and base rebar.
`<model>.tcvn.json`	The TCVN deflection definition.

All of them travel inside the `.optp` that you save.

12.5 5. What goes where

The ADAPT INP format cannot hold everything a SAFE file holds. OpenPT writes what it can into the INP part of the model and the rest into sidecars:

Item	Where it is kept
Planes, slab regions (thickness, E, ν, γ), openings, beams, design-only columns	INP
Load cases, one per SAFE pattern: DEAD/SUPERDEAD → Dead; LIVE → Live; WIND → lateral (Wind); QUAKE → lateral (Seismic); other → user case	INP
Point, line and area loads with all six components (FX, FY, FZ, MX, MY, MZ), including lateral and `*_ABOVE` loads	INP
Self-weight (the pattern's SELFWEIGHT multiplier)	INP (automatic self-weight × multiplier)
Linear-add combinations of plain pattern cases	INP LOAD_COMBINATIONS
Point supports and springs	`<model>.openpt.json`
Imposed displacements (POINTDISPL)	`<model>.openpt.json`
Linear static cases made of several patterns (for example SWL = SWL + SWL_ABOVE)	`<model>.openpt.json`
Modal case (number of modes, mass source)	`<model>.openpt.json`
Response spectrum cases (function, damping, directions, CQC/SRSS)	`<model>.openpt.json`
Cover values the INP cannot carry, beam releases with no INP block	`<model>.openpt.json`
Envelope, SRSS, nested and spectrum-based combinations	`<model>.combos.json`

OpenPT's analysis reads these sidecars, so the converted model gives the same reactions and displacements as SAFE's own analysis of the file (gravity, `*_ABOVE`, wind, modes, spectra and all combinations).

ADAPT-Builder does not read the OpenPT sidecars. When you export to ADAPT, OpenPT lists what ADAPT will not see, for example "ADAPT will not see: n imposed displacements, modal/RS cases ...". TCVN punching results are never written to `.adm` either.

Headless use: `OpenPT.Gui.exe --f2k-to-inp <in.f2k> <out.INP> [options]` (see "Command-line use").

12.6 6. Working with the imported model

A typical order:

1. **Check** the import report and the model (**Analyse** ▸ **Check** ▸ **Check Model**, F5).
2. **Edit** what you need: columns, walls, beams, openings, loads (the Model and Loads tabs).
3. **Mesh and analyse**: **Analyse** ▸ **Meshing** and **Analyse** ▸ **Run** ▸ **Run Analysis** (F9). The mesh cell comes from the import (600 mm); 500 mm is a good value for design.
4. Show a result in the **Results View** pane (chapter "Results"). The import report is on its **Report** tab.
5. **Design code**: choose it in **Design** ▸ **Criteria** ▸ **Design Code...** (TCVN 5574:2018 for Vietnamese practice). **Loads** ▸ **Define** ▸ **TCVN deflection...** makes the TCVN total deflection (chapter "TCVN 5574:2018 design").
6. **Strips**: **Design** ▸ **Strip Modelling** ▸ **Auto Strips...** and **Design** ▸ **Section Design** ▸ **Design the Sections**; then **Tendons & Rebar** ▸ **Generate** ▸ **Generate Rebar** and **Analyse** ▸ **Punching** ▸ **Shear Check** (chapter "Design").
7. **Save** as `.optp`, and export what you need (section 7).

The model is **locked** while analysis results exist (chapter "Files and projects"), as in SAFE.

12.7 7. Going back to SAFE, ADAPT and ETABS

- **File** ▸ **Export** ▸ **SAFE model (.F2K)...** writes a SAFE text file of the model (SAFE 20 and later table format; `--safe12` on the command line writes SAFE 12). **SAFE model (.FDB)...** writes a SAFE 22 database. Objects that SAFE cannot hold are not written; the export lists them ("unsupported objects") when it writes.
- **ADAPT model (.INP)...** and **ADAPT-Builder model (.adm)...** write the ADAPT files (ADAPT does not see the sidecar data above).
- **Report** ▸ **ETABS** (**Send to ETABS...**, **Get ETABS Forces...**, **Import ETABS Forces from File...**) sends a floor to ETABS and brings the forces back (chapter "Design", section "Sending a floor to ETABS").
- The results are written as SAFE-layout CSV files with **File** ▸ **Export** ▸ **Export step...** (the load chosen in the **Load:** box) and **Export all...** (every case, mode, spectrum case and combination).

12.8 8. Older SAFE-kind documents (legacy)

An `.optp` made by an older build can hold a document of the **SAFE kind**: a SAFE file that was opened as a read-only document and analysed with OpenPT's own mesh, without conversion. No command of this build creates such a document, but **Open** still opens it. It uses the same ribbon, with the groups that belong to it:

Where	Commands (SAFE-kind document only)
Model ▸ **Add Structural Components**, **Model** ▸ **Convert**	**Openings...** (a large button of Add Structural Components: load or unload the openings sidecar; while the model is locked it is disabled and its tooltip says "The model is locked because it has results, and the openings change its mesh. Unlock it (Lock/Unlock Model) to change the openings; the SAFE file itself is never changed."); **Convert to OpenPT Model** (Convert group). There is no **F2K** group in the Model tab

Where	Commands (SAFE-kind document only)
View ▸ Views, **Analyse ▸ Results**	the model, results, tables and strip plan views (the small buttons **Model**, **Results**, **Tables** and **Strip plan** of View ▸ Views); in the Results group **Results group**, **Result step**, **Slab component**, **Beams component**, **Clip shell range** and **Deformed shape (3D)**; **Export step...** and **Export all...** are in the File menu. The quantities have the names of an OpenPT model: Slab deflection, Displacement X/Y, Rotation about X/Y/Z, Moment Mx/My, Twisting moment Mxy, Shear Vx/Vy and Membrane force Nx/Ny/Nxy (beams: Beam moment, Beam shear, Beam axial force, Beam torsion, Beam lateral moment and Beam lateral shear); the status line and the 3D caption read "Slab deflection (mm)" and "Beam moment (kN·m)". The file's column ids (U3, M11 and the like) appear only in the headers of exported CSV files. The plan has no caption line over it: the title of the colour bar carries the name, then the load case, then the line of the diagram. Columns whose file ids look like C5ACAF are labelled C1, C2 ... on the plan (the file id is in the tooltip). After an analysis the plan shows the deflection contour alone (beam diagrams appear only when you pick a beam quantity), and the info line says "Analysed N load cases. Pick a result." (or "Analysed, but the equilibrium check failed: check the supports and the loads, then Analyse again.").
Design ▸ Design Options	**Concrete:** (B35), **Steel:** (CB500-V), **Strips (mm):** (2000), **Bar (mm):** (12), **Design (TCVN, uniform strips)**, **Punching reinforcement**, **Gravity patterns only**, **Auto Strips...** and **Design**
Analyse ▸ Meshing	**Cell size (mm):** (default 1200; 500 mm when the design is on)
Analyse ▸ Punching	**Punching bars...**
Loads ▸ Define	**Load Patterns** (a read-only list)

After a SAFE file opens, the info line reads "Opened K2HH1_T10.F2K (read-only). See the Import report." (the counts and sidecars are in its tooltip). The window title reads "OpenPT — .F2K (read-only)", and the file segment of the status bar reads " (read-only, locked)" while the model is locked. The Import report says that the slabs were meshed into shell elements on a given mesh and the beams into beam elements, and how many notes are in the technical details. The plan shows the grids, the dimensions in m and filled supports.

A SAFE-kind document cannot be edited (geometry, loads, sections); it is saved in the `.optp` with its sidecars and its results store, and its analysis takes from tens of seconds to about a minute. To use every command, import the original SAFE file again with **Import model...**.

12.9 9. Limitations of the SAFE path

- Only rectangular beam sections are read; any other section type stops the import with a clear error.
- The shell moments of OpenPT's plate element differ from SAFE's by a few percent of the maximum, more next to point supports and in slab slivers narrower than about 100 mm. Do not use element-corner shears V13/V23 as design shears.
- Loads on points that fall on no slab element are dropped (as SAFE does). Read the orphan-load warnings in the import report.
- A SAFE file must be one slab level; the converted model can be made multi-level with the Levels tools.
- Imposed displacements are not applied (section 1).
- The design tables are not part of the results store; they are made again by **Design the Sections**.

13 TCVN 5574:2018 design

This chapter describes the TCVN checks of OpenPT. They work on any model, whether built in OpenPT or imported from ADAPT or SAFE, when the design code of the document is **TCVN 5574:2018** (**Design ▸ Criteria ▸ Design Code...**; SP 63.13330.2018 uses the same checks with its own values):

1. the total (long-term) deflection F = f1 − f2 + f3 and its check;
2. flexural design of strips and design sections;
3. punching shear without shear reinforcement;
4. punching shear with calculated shear reinforcement.

The older SAFE-kind documents (chapter "Working with SAFE models", section "Older SAFE-kind documents") have their own buttons for the same checks; they are marked "legacy" below.

13.1 1. Total deflection F = f1 − f2 + f3

1.1 Method

This is the uncracked total deflection per TCVN 5574:2018 and TCVN 2737:2023, computed at every joint:

```
f1 = ks · r · (G + Q)          short-term deflection under the full load
f2 = ks · r · (G + η·Q)        short-term deflection under the long-term part
f3 = (1 + ϕ) · r · (G + η·Q)   long-term deflection under the long-term part, with creep
F  = f1 − f2 + f3
```

where:

- G, Q = downward displacement of the permanent and live cases, from the linear analysis;
- r = E_model / E_b, where E_model is the slab modulus in the model and E_b the modulus of the chosen TCVN class;
- ks = 1/0.85 (cracked stiffness factor of the short-term curvature, 0.85·E_b);
- η = long-term part of the live load (default 0.35);
- φ = creep coefficient φ_b,cr of the class and humidity (TCVN 5574:2018 table).

The formula is linear in the load cases, so F is an ordinary linear combination of cases:

```
factor on each G case:  a_G = r · (1 + ϕ)
factor on each Q case:  a_Q = r · (ks · (1 − η) + η · (1 + ϕ))
```

Example: B35, humidity 40–75 % (φ = 2.1), η = 0.35, r = 1. Then a_G = 3.1 and a_Q = 1.176 × 0.65 + 0.35 × 3.1 = 1.850.

TCVN values used by OpenPT:

Class	R_b (MPa)	R_bt (MPa)	E_b (MPa)	φ (>75 %)	φ (40–75 %)	φ (<40 %)
B25	14.5	1.05	30000	1.8	2.5	3.6
B30	17.0	1.15	32500	1.6	2.3	3.2

Class	R_b (MPa)	R_bt (MPa)	E_b (MPa)	φ (>75 %)	φ (40–75 %)	φ (<40 %)
B35	19.5	1.30	34500	1.5	2.1	3.0
B40	22.0	1.40	36000	1.4	1.9	2.8
B45	25.0	1.50	37000	1.3	1.8	2.6

1.2 The TCVN deflection… button (recommended)

1. Run the analysis (**Analyse ▸ Run ▸ Run Analysis**, F9). Before an analysis the button says "TCVN deflection needs an analysis first: run the analysis (F9)."
2. Press **Loads ▸ Define ▸ TCVN deflection…** The dialog "TCVN total deflection" opens. It has six sections:

1. Load cases. One row per linear static case: Case | Patterns (type) | Class | Factor. The class (G, Q or —) is proposed from the types of the patterns in the case: - all DEAD or SUPERDEAD → **G**; - all LIVE or reducible live → **Q**; - anything else (WIND, QUAKE, OTHER, mixed, modal, spectrum) → **—** (not used).

You can change the class and the factor of any row.

Take the cases from combination: (default "(none)"). When you choose a combination, its cases are classed by their pattern types and its factors multiply the case results; the other cases are set to —. A case at factor 0.9 enters as 0.9 × case. The dialog proposes the first service combination that holds every G and Q case with factor 1, or says that there is none. Lateral cases are dropped by their class.

2. Loads from above. "Exclude loads from the storeys above (patterns named *_ABOVE)", default **on**. The `*_ABOVE` patterns that ETABS writes carry imposed support displacements of about 20 mm, which would distort the deflection. When ticked, the G and Q cases are taken from a second run without those patterns. The run is made once and kept with the analysis; the status line says when it is solving.

3. Parameters. - **Concrete:** B25–B45. The default is the class whose E_b is nearest to E_model, shown as "(E_b nearest to E_model: …)". - **Humidity %:** >75, 40–75 (default) or <40. - **eta:** 0.35. - **ks:** 1/0.85 = 1.1765, editable. - φ, E_model and E_b are shown.

4. Preview. The factors a_G, a_Q, r and φ, and the combination text, in the form `TCVN_F = a_G·SWL + a_G·FNS + … + a_Q·LL…`, with the numeric factors of your model.

5. Names. Combination name: (default `TCVN_F`) and "also create TCVN_f1, TCVN_f2, TCVN_f3" (default on).

6. Limit (optional). Allowable deflection (mm): or **L (mm):** with **/ x:** (for example L = 8000, x = 250). Empty by default.

3. Press **OK**. The combinations are created and `TCVN_F` is selected in Results with the deflection contour (**Slab ▸ Deformation ▸ Slab deflection**). Re-running the dialog with the same name replaces the combination. **Delete** removes the combinations and the definition file. **Cancel** does nothing.

1.3 Results and check

- `TCVN_F` (and `TCVN_f1`, `TCVN_f2`, `TCVN_f3`) behave like any combination: contour, values, tables, Export step.

- The info line reports F max (downward, mm) with its joint and x, y.
- If a limit was entered, the info line also gives "OK" or "NOT OK" and the ratio F max / limit (for a limit of L/250 on an 8000 mm span, the limit is 32 mm).
- The TCVN deflection table lists the 25 joints with the largest F, with columns Joint, x, y, G, Q, f1, f2, f3, F (mm). `tcvn_uz.csv` holds every joint: joint, x, y, G, Q, f1, f2, f3, F.
- The definition (classes, options, parameters, names, limit) is saved in `<model>.tcvn.json` (inside the `.optp`). After the next analysis, re-opening re-creates the combinations from this file.

Which limit to use. OpenPT does not impose a limit. Enter the allowable value yourself: L/250 per the usual practice, or the limit of TCVN 2737:2023 for the span and the case, with L the span you are checking. The program checks the single largest F over the whole floor against the one limit entered. If spans differ, check each panel against its own span.

1.4 Deflection in the design run

Design ▸ Section Design ▸ Design the Sections with the code TCVN also computes the total deflection and the TCVN deflection check (a `tcvn_deflection.csv` next to the other design outputs); the beams of the **Beam** tab have the same check (see "Beam design"). The run takes G and Q from the cases by their type, uses η = 0.35 and φ from the concrete class and the humidity of the criteria (40–75 % by default), and never uses the `*_ABOVE` patterns. For other class assignments, use **TCVN deflection...** above.

For a legacy SAFE-kind document, **Analyze** with **Design (TCVN, uniform strips ...)** ticked computes F in the same way, with G = the cases named SWL, FNS, MRS or MSR and Q = the cases whose name starts with LL, as a second, gravity-only run; the result is the "TCVN F" contour of **Results:** and the **TCVN deflection** tab of the Tables view.

1.5 Limitations

- The method is the uncracked total deflection with ks = 1/0.85 standing in for short-term cracking. It is not a cracked-section analysis. For PT and RC slabs where cracking matters, use **Analyse ▸ Run ▸ Cracked Deflection** (chapter "Design").
- After results are loaded from a results store, the TCVN combinations are **not** recreated automatically, because they need the run without loads from above. Press **TCVN deflection...** again and OK.

13.2 2. TCVN flexural design of strips and sections

Applies to the design strips of **Auto Strips** and to drawn support lines (chapter "Design"), on any model, and to the uniform strips of a legacy SAFE-kind document. Prestressed strips (tendons) are designed as well: bonded tendons at R_sp, unbonded at 0.9 σ_sp with no increase, tendons in the compression half left out (spec 165).

Materials. Concrete: B25–B45 (default B35), **Steel:** CB400-V (R_s = 350 MPa) or CB500-V (R_s = 435 MPa, default). With B35 / CB500-V: R_b = 19.5 MPa, R_s = 435 MPa.

Effective depth (covers from **Tendons & Rebar ▸ Rebar Criteria ▸ Rebar Cover / Size**, default 25 mm, and the preferred bar size; on a legacy SAFE-kind document from the slab section, CTI/CBI, and the **Bar (mm):** box, default 12):

- Layer A (X strips, outer layer): h0 = h − clear cover − bar/2
- Layer B (Y strips, inner layer): h0 = h − clear cover − bar − bar/2

Flexure (TCVN 5574:2018 8.1.2, rectangular block, single reinforcement), for each section piece and each strength step:

```
α_m = M / (R_b · b · h0²)
ξ   = 1 − √(1 − 2·α_m)
A_s = R_b · b · ξ · h0 / R_s
ξ_R = 0.8 / (1 + ε_s,el / ε_b2),   ε_s,el = R_s / E_s
```

- Bottom steel comes from the largest sagging moment, top steel from the largest hogging moment.
- If ξ > ξ_R, the face is flagged **over-reinforced**. The tension steel is given at ξ_R, and the compression steel the section would need is reported for information (`As_comp_top`, `As_comp_bot`).

Minimum steel (10.3.6): μ_min = 0.1 %, so A_s,min = 0.001 · b · h0. In the "with minimum" figures, A_s = max(A_s,req, A_s,min) at stations where the face needs steel.

Rebar mass = Σ A_s × 250 mm × 7.85e-6 kg/mm³ over the stations of each strip and face. It is given "strength only" and "with minimum", per layer and face, and in total.

`strip_design.csv` **columns:** layer, strip, station, x, y, slab_section, piece_width, h, d_top, d_bot, M_neg_design_Nmm, neg_combo, M_pos_design_Nmm, pos_combo, As_top_req, As_bot_req, As_min_top, As_min_bot, As_top, As_bot, over_top, over_bot, As_comp_top, As_comp_bot (N, mm, mm² per piece width).

Limitations. Section actions are slab-only: beams are not included and are designed elsewhere. M12 is ignored for generated (axis-parallel) strips. No crack-width check is made in this design.

13.3 3. TCVN punching shear without reinforcement (8.1.6)

Checked at every column of the model by **Analyse ▸ Punching ▸ Shear Check** and by **Design the Sections** (the columns made on import, or those of the columns sidecar of a legacy SAFE-kind document).

3.1 Geometry

- **h0** = h − top cover − bar: the mean effective depth of the two layers (h − c − bar/2 and h − c − 3·bar/2). h and the cover come from the slab section at the column centre. If a slab section has no CTI, a top cover of 25 mm is assumed and a warning is written.
- **Design contour**: the rectangle at h0/2 from the column faces.
- **Edge and corner columns**: the parts of the contour outside the slab are dropped (open contour, with its own centroid; moments are taken about it).
- **Openings**: within 6·h0 of the column face, the part of the contour between the tangents from the column centre to the opening is removed, as is any part inside an opening. The removed length is reported.
- **R_bt** from the class on the design bar (B35: 1.30 MPa).

3.2 Check

```
F_b,u  = R_bt · u · h0
M_bx,u = R_bt · W_bx · h0          M_by,u = R_bt · W_by · h0
f  = F / F_b,u
m  = Mx / M_bx,u + My / M_by,u
r0 = f + min(m, 0.5·f)             OK when r0 ≤ 1
```

W_bx and W_by are the section moduli of the contour line about its centroidal axes (smaller of the two sides). For a circular column, $W_b = \pi(D + h0)^2/4$.

Forces. F is the slab's action on the column (support reaction plus direct point loads at the column points). The load inside the contour is **not** subtracted (conservative).

Steps and verdicts. Every strength step is checked:

- **Status** = the verdict over all ULS steps;
- **Status no RS** = a second verdict over the steps without a response spectrum term.

The governing step is the one with the largest ratio. The Punching table of a legacy SAFE-kind document shows the verdict over all ULS steps. It can show the one without a response spectrum term instead (section 4.6).

13.4 4. TCVN punching shear with calculated reinforcement (8.1.6.2.2, 8.1.6.3.2)

4.1 Formulas

```
q_sw   = R_sw · A_sw / s_w                (128)
F_sw,u = 0.8 · q_sw · u                   (127)
M_sw,u = 0.8 · q_sw · W_sw                (133)
```

Condition (132):

```
F/(F_b,u + F_sw,u) + Mx/(M_bx,u + M_swx,u) + My/(M_by,u + M_swy,u) ≤ 1
```

with:

- the moment part ≤ 0.5 × F/(F_b,u + F_sw,u);
- caps: F_b,u + F_sw,u ≤ 2·F_b,u, M_bx,u + M_swx,u ≤ 2·M_bx,u, M_by,u + M_swy,u ≤ 2·M_by,u;
- **the 0.25·F_b,u rule:** the reinforcement counts only if F_sw,u ≥ 0.25·F_b,u. OpenPT therefore never provides less than this.

R_sw (TCVN 5574:2018 table 14):

Punching bar grade	R_sw (MPa)
CB240-T	170
CB300-T, CB300-V	210
CB400-V	280

Punching bar grade	R_sw (MPa)
CB500-V	300

The punching bar grade is a separate setting, **not** the main bar grade. Its default is **CB300-V**.

γ_b1 (default 1.0) multiplies R_bt. Use 0.9 for long-term loads (6.1.2.3).

4.2 Layouts

Uniform (Fig. 18), the default. Bars are spread evenly along the contour, so W_sw = W_b. Every sum is then (1 + k) times the concrete term, with k = 0.8·q_sw/(R_bt·h0). The required k has a closed form:

```
k_req     = max(r0 − 1, 0.25)
q_sw,req  = k_req · R_bt · h0 / 0.8
a_sw,req  = q_sw,req / R_sw              (mm²/mm, reported per metre of contour)
```

Because of the 2× caps, k ≤ 1. A column with **r0 > 2** therefore cannot be reinforced: its status is "Exceeds 2·F_b,u".

Cross (Fig. 15d). Studs lie on the column axes only.

- u_sw = L_sw,x + L_sw,y, the contour length covered by the bars, and F_sw,u = 0.8·q_sw·u_sw.
- W_sw is computed from those lengths by the same rule as W_b.
- q_sw,req is found by bisection on condition (132), with all caps.
- The outer contour for the concrete is the diagonal contour of Fig. 15d.
- A circular column, or a contour that the cross strips do not reach, falls back to the uniform layout, with a note.

Edge and corner columns (8.1.6.1). Two variants are checked:

- **open**: the contour to the slab edge;
- **closed**: the contour round the column, only when it lies wholly inside the slab.

The governing one (the larger ratio) is used and reported in the Variant column.

4.3 Detailing (10.3.4.7)

- Row spacing perpendicular to the contour: s_w ≤ h0/3 and ≤ 300 mm. **Auto** takes the largest multiple of 5 mm that meets both.
- First row from the column face: between h0/3 and h0/2. **Auto** is h0/3 rounded down to 5 mm.
- The reinforced zone extends at least **1.5·h0** from the column face.
- Leg spacing along the contour: s_t ≤ 1/4 of the contour side and ≤ 300 mm.
- Practical minimum leg spacing: 50 mm. If less is needed, the bar diameter steps up 10 → 12 → 14 → 16 mm. If Ø16 still needs less than 50 mm, the status is "too dense".
- Option **Allow s_w up to h0/2**: only the bars crossing the punching pyramid count (8.1.6.2.2, last paragraph), and the notes say so.
- Anchorage of the bars (10.3.4.9) is **not checked**. A note says so.

Spacings you enter that break these rules are noted in the notes column (for example "s_w … mm exceeds the limit of 10.3.4.7").

4.4 The design procedure (per column)

1. Compute r0 for the governing step. If r0 ≤ 1, the status is **"No reinforcement needed"**.
2. Uniform layout with r0 > 2, or cross layout that cannot satisfy (132) at the 2× caps: the status is **"Exceeds 2·F_b,u: increase h, column size or concrete class"**.
3. Compute q_sw,req and a_sw,req.
4. **Layout.** n_band is the number of rows inside the band of width h0 centred on the contour (rows at a1, a1 + s_w, …). The leg spacing is s_t = min(n_band·A_bar/a_sw,req rounded down to 5 mm, side/4, 300 mm). The diameter is stepped up as needed. Then the provided q_sw = R_sw·n_band·A_bar/s_t, and condition (132) is re-checked with it. The status is **"Reinforcement provided"**, with the ratio.
5. **Zone extent.** Rows are added at s_w until both of these hold: - the outer contour, 0.5·h0 beyond the last row, passes the no-reinforcement check with the same F, Mx and My (and its own u, W, openings and edge clipping); - the last row is at least 1.5·h0 from the face.

If **Most rows** is reached first, the status is **"Outer contour fails"**. 6. **Studs and stirrups.** Studs sit on rails perpendicular to the column faces, at s_t along the contour and s_w along each rail. Stirrups are closed links at the same positions. The count is the same; only the label differs.

4.5 Options

With the design code TCVN or SP 63, **Analyse ▸ Punching ▸ Shear Design Options** has no separate TCVN button and no separate punching code: the design code decides (an older model's own punching code is shown as "Punching code (legacy)" with a remove button). The reinforcement is set in the **Reinforcement** group of the dialog: **Studs** or **Stirrups**, bar Ø, N_Ds, N_Dr and α. The other options of the table below keep their defaults.

A legacy SAFE-kind document keeps the dialog "Punching reinforcement (TCVN 5574:2018)", opened with **Analyse ▸ Punching ▸ Punching bars…**; its settings are saved in the design sidecar of that document. Its options are:

Option	Choices	Default
Layout	Uniform (Fig. 18), Cross (Fig. 15d)	Uniform
Bars	Stirrups (closed links), Studs (rails)	Stirrups
Bar Ø (mm)	8, 10, 12, 14, 16, 20 (start of the step-up)	10
Grade	CB240-T, CB300-T, CB300-V, CB400-V, CB500-V	CB300-V
s_w (0 = auto)	mm	0 (auto)
First row (0 = auto)	mm	0 (auto)
γ_b1	above 0, at most 1.2	1.0
Most rows	1–200	20
Allow s_w up to h0/2	on/off	off

Defaults restores these values. Settings take effect at the next **Analyze**; the info line reminds you.

4.6 Running it on a legacy SAFE-kind document

1. The columns sidecar is read with the document.

2. Tick **Design (TCVN, uniform strips ...)** and **Punching reinforcement** in **Design ▸ Design Options**. Set the bars with **Analyse ▸ Punching ▸ Punching bars...**.
3. Press **Analyze** (F9). The info line summarises the run, for example "punching reinforcement (TCVN 8.1.6.2.2, Uniform layout, all ULS steps): n OK, n reinforced, n exceed 2 F_b,u, n outer contour fails, n too dense; max r0 ..., punching bars ... kg".
4. Look at the plan:

Colour	Meaning
Green	No reinforcement needed.
Orange	Reinforcement provided.
Red	Exceeds 2·F_b,u, outer contour fails, or too dense.

Show punching reinforcement draws the legs and the outer contour at the reinforced columns.

Labels. Beside each orange column, next to its name, the plan writes the reinforcement as `Ø10 CB300-V · 4 rows · sw 70 · st 150 · 12.4 kg` (bar diameter, grade, rows, s_w, s_t, mass). When the view is zoomed out far enough, the label is split in two lines after the rows. The labels follow the **Labels** row of **View ▸ Layers ▾**. Its default is off, so ticking **Show punching reinforcement** alone draws the legs and the contour but no labels. A column that the filter **Punching: only columns with no beams deeper than the slab** fades has no label. 5. Open **Tables ▸ Punching** for the numbers, and **Export tables...** for the CSV files.

The design is made for the governing ULS step (largest r0) and also, separately, for the governing step without a response spectrum term. Both exist after one **Analyze**.

Which design the table and the plan show. The check box **Plan colours: steps without a response spectrum term** (Tables view) chooses:

- Unticked: the column colours, the drawn legs and labels, and the Punching table all use the design over all ULS steps.
- Ticked: all three use the design over the steps without a response spectrum term. You do not need to analyse again to switch.

The switch changes the Punching table only when the run has a response spectrum step and a design without it exists. If not, the table stays as in the unticked state.

Punching table columns (Tables view), box unticked:

- Without reinforcement: Column, bx, by, Position, Beams in, Deep beams in, h, h0, u, Governing combo, N kN, Mx kNm, My kNm, Fb ult kN, Ratio, Status, Ratio no RS, Status no RS, Removed by openings mm.
- With reinforcement: Reinforcement, k req, a_sw req mm2/m, Bar dia, s_w, First row, s_t, Legs row 1, Rows, Zone width, Total legs, Mass kg, Ratio with steel, u out, Ratio out, Variant, Reinforcement no RS, Mass no RS kg.

Box ticked: the columns Governing combo, N kN, Mx kNm, My kNm, Ratio and Status, and every reinforcement column from Reinforcement to Variant, show the no-RS values under headers with the suffix " (no RS)", for example **Ratio (no RS)** and **Mass kg (no RS)**. The four columns Ratio no RS, Status no RS, Reinforcement no RS and Mass no RS kg are not shown, because they would repeat. The other

columns (Column, bx, by, Position, Beams in, Deep beams in, h, h0, u, Fb ult kN, Removed by openings mm) do not change.

Rows are shaded green for OK and red for FAIL, by the status column that is shown.

`punching.csv` (one row per column, governing step). The base columns are:

- column, bx, by, position, area, h, h0, u;
- governing_combo, N_slab, Mx, My, Fb_ult, Mbx_ult, Mby_ult, F_ratio, M_ratio, ratio;
- N_reaction, ratio_reaction;
- governing_no_spectrum, ratio_no_spectrum, status_no_spectrum;
- removed_by_openings, removed_by_slab_edge, contour_full, centroid_x, centroid_y, status.

With reinforcement on, these columns are added:

- reinf_status, reinf_step, r0, r_with, k_req, qsw_req_N_mm, asw_req_mm2_m;
- bar_dia_mm, bar_grade, sw_mm, first_row_mm, st_mm, legs_x, legs_y, legs_row1, rows, zone_width_mm, total_legs, mass_kg;
- u_out_mm, r_out, reinf_variant, reinf_notes;
- reinf_status_no_spectrum, r_with_no_spectrum, rows_no_spectrum, total_legs_no_spectrum, mass_kg_no_spectrum.

`punching_all.csv` (every column × step) has these columns: column, step, N_slab, Mx, My, F_ratio, M_ratio, ratio, moment_capped, N_reaction, ratio_reaction, note. With reinforcement on, it adds reinf_status, k_req, asw_req_mm2_m and variant.

4.7 TCVN punching on an OpenPT model

On any OpenPT model (built in OpenPT, or imported from ADAPT or SAFE), the TCVN check uses the same calculator, contours and supports as the EC2/ACI punching check, and runs when the design code is **TCVN 5574:2018** (or SP 63.13330.2018).

1. Choose the code in **Design ▸ Criteria ▸ Design Code...**, and the reinforcement in **Analyse ▸ Punching ▸ Shear Design Options** (section 4.5).
2. **Analyse ▸ Punching ▸ Shear Check**. Results View ▸ Punching Shear shows the statuses (**Stress check**, **Stress ratio**, **Condition**, **Reinforcement**) and the plan shows the markers. **Design the Sections** and **Tendons & Rebar ▸ Generate ▸ Generate Rebar** run the same check.
3. To see the bars, open the **Results View** pane (**Analyse ▸ Results ▸ Results Display Settings**). In the **Load:** drop-down next to the **Plan** tab choose the **Envelope** or one combination. On the **Analysis** tab tick **Punching Shear ▸ Reinforcement layout**; the **Rebar Display Manager** option **Display punching reinforcement** shows the calculated bars together with the rebar.

Reinforcement layout. At every support where the TCVN check gave reinforcement (status "Reinforcement provided" or "Outer contour fails"), the Plan view draws the legs of the rows (small circles), the outer contour (dashed) and a label such as `Ø10 CB300-V · 4 rows · sw 70 · st 150 · 12.4 kg`. The label is on one line and is not tied to the Labels layer. Supports with no reinforcement, and the EC2 and ACI results (which have their own label, see "Punching shear" in the chapter "Design"), are not affected.

TCVN punching is not written to `.adm` files.

4.8 Hand check

For an interior 400 × 400 column, h0 = 210 mm, B30 (R_bt = 1.15 MPa), axial force only:

- u = 4 × (400 + 210) = 2440 mm, so F_b,u = 1.15 × 2440 × 210 = 589 kN.
- If F = 766 kN, then r0 = 1.30 and k_req = 0.30, so q_sw,req = 0.30 × 1.15 × 210 / 0.8 = 90.6 N/mm. With CB300-V (R_sw = 210 MPa), a_sw,req = 0.431 mm^2/mm = 431 mm^2/m of contour.
- If F = 648 kN (r0 = 1.10), the 0.25 minimum governs: k_req = 0.25.
- If r0 = 2.1, the status is "Exceeds 2·F_b,u".

4.9 Known limitations

- **Punching table and `punching.csv`:** the table follows the box **Plan colours: steps without a response spectrum term** (section 4.6). `punching.csv` does not: it always holds both designs, the all-steps columns and the `*_no_spectrum` columns.
- **Default bar grade:** the punching bar grade defaults to **CB300-V** (R_sw = 210 MPa), independent of the main steel (CB500-V by default). Check that this matches your drawings and change it in the Reinforcement options of the punching dialog if needed.
- The load inside the contour is not subtracted from F (conservative).
- Anchorage (10.3.4.9) is not checked.
- SAFE 20 gives "Not Calculated" for TCVN/SP 63 punching, and ADAPT-Builder has no TCVN punching, so there is no external program to compare with. The method was checked by hand calculations.
- **TCVN combinations after loading a results store** need the **TCVN deflection...** button pressed again (section 1.5), and the design tables need **Analyze**.

14 Worked example: a SAFE floor from import to TCVN design

This walk-through takes one floor exported from ETABS, `MyTower_L10.F2K`, through import, analysis, TCVN deflection, slab rebar and punching with reinforcement, in the one-ribbon program. The numbers you see depend on your model; the steps do not.

14.1 Step 0. Prepare the folder

```
MyTower_L10\
    MyTower_L10.F2K               ETABS ▸ Export ▸ SAFE .F2K (floor 10)
    MyTower_L10.openings.json     optional: slab openings
    MyTower_L10.columns.json      optional: column rectangles and their points
```

The sidecars must have the same stem as the F2K. Their coordinates are the model coordinates of the file (mm unless `units` says otherwise). Without a columns file the import makes a column at every restrained point, of the size DIMX × DIMY of the point.

14.2 Step 1. Import the file

1. **File ▸ Import ▸ Import model...** and choose `MyTower_L10.F2K`.
2. OpenPT converts the file without a dialog. The info bar says what it made; the **Check Model** window and the **Report** tab of the Results View pane hold the import report, and `MyTower_L10.import.txt` is written next to the file.
3. Read the report: the counts, the walls and columns made, and "What was skipped, and what to do". Imposed displacements (`POINTDISPL`) are listed there and are not applied.
4. **File ▸ Save As** and keep `MyTower_L10.optp`. Everything below is saved in it.

14.3 Step 2. Check the model

1. Look at the Plan (the start view is the structural plan) and, with **View ▸ Views ▸ View ▸ Plan + 3D**, at the 3D view. Walls and columns are real members below and above the slab.
2. **Analyse ▸ Check ▸ Check Model** (F5) lists the messages of the model.
3. Choose the design code: **Design ▸ Criteria ▸ Design Code...** and pick **TCVN 5574:2018**. The concrete class, the steel grade and the humidity are in **Design ▸ Criteria ▸ Materials** and the criteria dialogs (chapter "Loads, criteria and combinations").

14.4 Step 3. Mesh and analyse

1. **Analyse ▸ Meshing**: set the cell size. Use 1200 mm for a quick look and 500 mm for design.
2. **Analyse ▸ Run ▸ Run Analysis** (F9). The status bar shows the step, the percentage and the time, with a **Cancel** button.
3. When it finishes, the model is locked, the Results View pane opens, and the Plan shows the deflection contour of the first service combination. Pick any other load in the **Load:** box.

14.5 Step 4. Check the analysis

1. The **Summary** and the reactions are in the Results View and in the Analysis Data menu of the Analysis Reports group on the **Report** tab (Equilibrium Check, Solution Statistics).
2. Compare the base reactions with SAFE's: they agree to better than 1e-6 on a tower floor.

14.6 Step 5. TCVN total deflection

1. **Loads ▸ Define ▸ TCVN deflection...**. The dialog proposes the cases and classes (G, Q), the concrete class, humidity 40–75 % and η = 0.35.
2. Keep **Exclude loads from the storeys above (patterns named _ABOVE)* ticked, enter the limit (for example L = 8000 mm and x = 250), and press **OK**.
3. `TCVN_F` is shown with the deflection contour (**Slab ▸ Deformation ▸ Slab deflection**). The info line gives F max and "OK" or "NOT OK".

14.7 Step 6. Slab reinforcement

1. **Design ▸ Strip Modelling ▸ Auto Strips...**: give the strip spacing (2000 mm) and answer the question about existing lines. Or draw support lines with **Create X Support Line** along the column lines.
2. **Design ▸ Section Design ▸ Design the Sections**. It generates the sections if needed, designs them to TCVN 5574:2018 and opens the **Reinforcement ▸ Calculated (required)** result.
3. **Tendons & Rebar ▸ Generate ▸ Generate Rebar** draws the bars (**Rebar Options...** first, to change the load, the bar lengths and the orientation).
4. **Design ▸ Section Design** results are in the Results View under Design Sections; the strip design table is exported with **File ▸ Export ▸ Export Design Table (CSV)...**.

14.8 Step 7. Punching shear with reinforcement

1. **Analyse ▸ Punching ▸ Shear Design Options**: choose **Studs** or **Stirrups**, the bar Ø and the number of legs.
2. **Analyse ▸ Punching ▸ Shear Check**. Green columns need no reinforcement, orange ones get reinforcement, red ones exceed the code.
3. In the Results View tick **Punching Shear ▸ Reinforcement layout** to see the legs, the contour and the label of each column.
4. Check one column by hand with the formulas of section 4.8.

14.9 Step 8. Report and save

1. **File ▸ Export ▸ Calculation Report** writes the Word report (`PLTT_san_MyTower_L10.docx`).
2. **File ▸ Save** (Ctrl+S) saves everything in `MyTower_L10.optp`: the model, the sidecars and the results. Copy that one file to move the project.
3. After **Unlock** (which deletes the results), run the analysis again before you design.

14.10 Step 9 (optional). Continue in ADAPT, SAFE or ETABS

- **File ▸ Export ▸ ADAPT model (.INP)…** or **ADAPT-Builder model (.adm)…** continues the floor in ADAPT-Builder (it does not see the OpenPT sidecars).
- **File ▸ Export ▸ SAFE model (.F2K)…** gives SAFE a copy; **Report ▸ ETABS** sends the floor to ETABS and brings the forces back.

15 Known limitations

These limitations are known in this build. Check your results against them.

Area	Limitation	What to do
TCVN punching with spectrum steps	Response-spectrum steps (CQC/SRSS envelopes) are taken with their full absolute values, so their ratios are very large. It is normal for every column to show "Exceeds" when they are included.	Design punching for the non-spectrum steps, and treat the seismic case separately.
TCVN combinations after loading a store	The TCVN total-deflection combination is not rebuilt when a results store is loaded.	Press the TCVN combination button again after opening a solved model.
Converted SAFE beam line loads	After converting `.F2K` to `.INP`, beam line loads differ from SAFE by about 1.6e-5 (relative).	None needed; this is the size of rounding.
Drawn strips on SAFE models	The rebar mass from strips you draw yourself can be about 8 % below that of the automatic uniform strips: section cuts that pass over columns and walls are skipped.	Compare both when the mass matters.
Calculated rebar plan	It can still be generated while the model is locked: it is treated as read-only output. It is refused in Multi-Level mode and when a load case was skipped.	Switch to single-level mode, or run all load cases.
Imposed displacements	Support settlements or imposed displacements in an imported SAFE file are not applied in the analysis.	The import report lists them; model them by hand if they matter.
Time-dependent analysis	Every vertex is a pin or roller (no column rotational springs), rebar layers are not offered, and unbonded tendons use an average strain.	Read the limits printed in the report.
Reports	The text of the single default reports and the Report Compiler is English only.	None.
`.adm` export	TCVN punching results, TCVN combinations and SAFE-only data are not written to `.adm`. The export lists what is left out.	Keep the `.optp` or the results store as the full record.
Command-line runs	Self-tests and other command-line runs no longer add to **Recent Files** or write `%APPDATA%\OpenPT\error.log`, except a picture or analysis switch that keeps no settings folder of its own (for example `--render-plan`, `--render-3d`, `--analyze`): if it fails, its error is written to that log.	None needed; the log entry only records the failure.

16 Glossary

Term	Meaning
ADAPT-Builder	Commercial PT slab program whose `.INP` / `.adm` model files OpenPT reads and writes.
Balanced load	The equivalent load of the tendons (upward in spans, downward at high points).
CQC / SRSS	Modal combination rules of a response spectrum case; results are unsigned envelopes.
Design strip	A band of slab whose actions are integrated across its width at design sections.
Design section	A cut across a strip where the moment, shear and axial force are summed and designed.
F2K	SAFE's text export format.
f1 − f2 + f3	TCVN 5574:2018 total deflection: short-term deflection under the full load, minus the short-term deflection under the long-term part, plus the long-term deflection under the long-term part.
F_b,u, M_b,u	Punching capacity of the concrete (TCVN 8.1.6): R_bt·u·h0, and R_bt·W_b·h0.
Hyperstatic action	The secondary moments and reactions caused by prestress in an indeterminate structure.
Lock	The state of a model that has analysis results. Its geometry and loads cannot be changed until it is unlocked; unlocking discards the results.
`.optp`	OpenPT single-file project: a ZIP archive that holds the model, its sidecars and its results store.
q_sw	Punching reinforcement force per unit length of contour: R_sw·A_sw/s_w.
r0	TCVN punching ratio without reinforcement; 1.0 means fully used.
Results store	The folder `<model>_OpenPT\` next to the model, which holds the saved analysis results.
Sidecar	A small JSON file next to a model that holds OpenPT data the source format cannot hold (strips, columns, combinations, options).
Support line	A line along which design strips and design sections are generated.
Beam tab	The last ribbon tab, for beams and frames without a slab; it holds beam loads, analysis and design.
Section Check	A calculator (Design ▸ Section Design) that checks one section with forces you type, with no analysis; SP 63 and TCVN 5574:2018 (33 kinds), EC2 and ACI 318-14 / 318-19 (6 kinds).

17 Standards referenced

- TCVN 5574:2018: Concrete and reinforced concrete structures, design standard.
- TCVN 2737:2023: Loads and actions.
- EN 1992-1-1 (Eurocode 2).
- ACI 318-14 and ACI 318-19.
- SP 63.13330.2018 (the Russian code on which TCVN 5574:2018 is based; also offered as a design code).
- ACI 209R-92 (creep and shrinkage, time-dependent analysis).