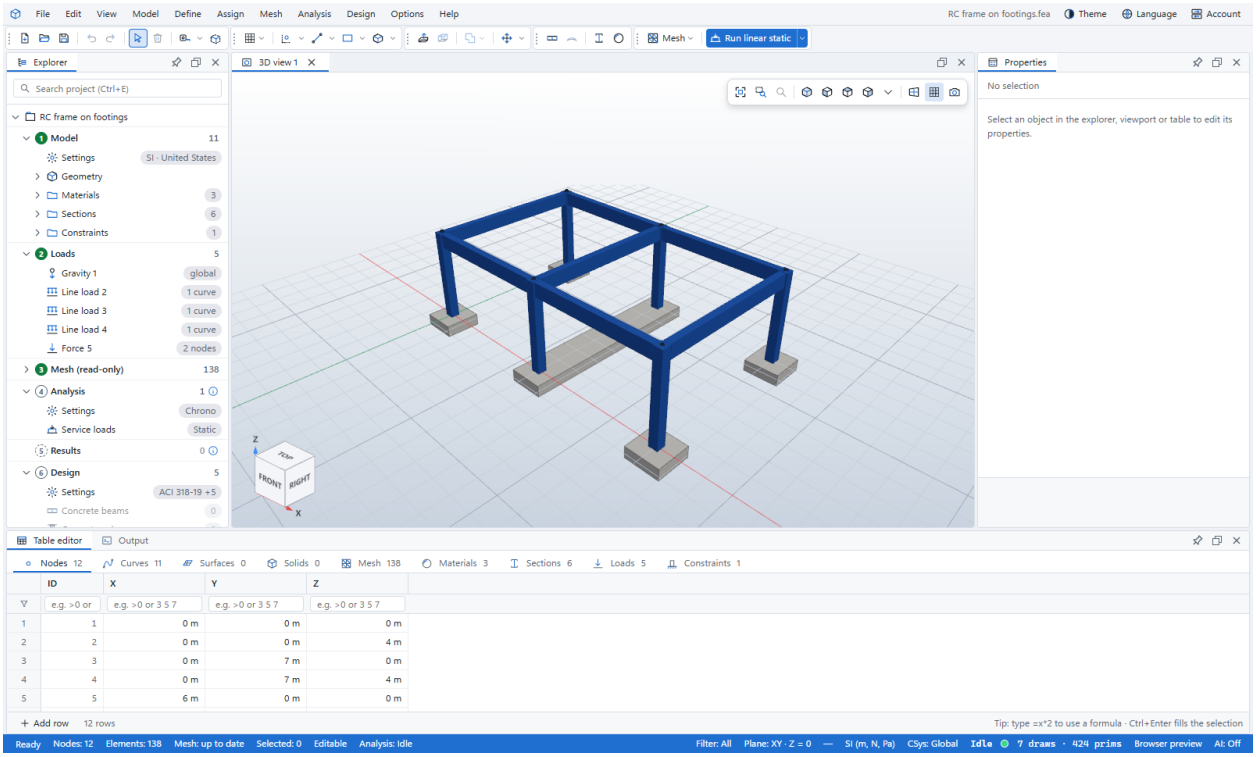
FEA STUDIO

User Manual

Part 1 of 5 · Modeling

Start a project, build the structural model, and give it materials, sections, supports, loads and a mesh.



The manual set

Part 1 Modeling (this book)
Part 2 Analysis
Part 3 Design
Part 4 Drawings
Part 5 Reporting

Edition November 2026 · describes FEA Studio as built from the current source

Contents

About this manual 4
Conventions 4
The workspace 5
The Explorer stages 5
Menus 6
Toolbars 7
Panels and layouts 8
Starting a project 9
The Start dialog 9
New model: region, units and design codes 10
Templates 11
Opening, importing and saving 12
Project information 13
Navigating the 3D view 14
Mouse and view cube 14
View commands 14
Clipping box 14
Selecting 15
Right-click menu 15
Drawing the geometry 16
Structural grid 16
Work plane 16
Points and curves 17
Surfaces 17
Typing exact values 17
Inference snapping 17
Solids 18
Push/Pull, Move, Rotate and Scale 18
Groups, clipboard and undo 19
The Table editor 19
Materials and sections 20
The Define menu 20
Materials 20
Sections (profiles) 20
Assigning sections to curves 21
Assigning shells to surfaces 22
Supports 23
Loads 24

Load cases 24
Applying loads 24
Load generators 25
Load combinations 26
Meshing 27
Mesh settings 27
Mesh commands 27
Mesh quality 28
Locked models 29
Settings, shortcuts and help 30
Settings 30
Keyboard shortcuts 30
Guided workflows 30
AI assistants 30
Desktop and browser 31
Checklist before analysis 32

About this manual

FEA Studio is a structural analysis and design application. It covers buildings, bridges, towers, roofs and shells, steel connections, foundations and reinforced concrete members. The user manual comes in five parts, one for each stage of the work:

Part	Covers
1 · Modeling	Starting a project, the workspace, geometry, materials, sections, supports, loads and meshing
2 · Analysis	Studies and the five analysis types, load combinations, the mesh and solvers, running, and reading the results
3 · Design	Code checks and member design for concrete, steel, composite, walls, footings and connections
4 · Drawings	Steel and reinforced concrete shop drawings, drawing standards and export
5 · Reporting	Model, analysis, design and calculation reports; print, PDF, Word and other outputs

This part describes everything you do before the first analysis.

Conventions

- Menu commands are written as **Menu › Group › Command**, for example **Model › Curves › Line**.
- Keys are shown like `Ctrl+S` or `F5`.
- Labels in **bold** are the words you will see on the screen.
- Unless a step says otherwise, the desktop app and the browser version work the same way. Differences are listed in "Desktop and browser" at the end of this part.

Note FEA Studio stores values in SI units internally. You can enter and read values in the unit system of the project, and you can type a unit with a value (for example `2500 mm`) in most number fields.

The workspace

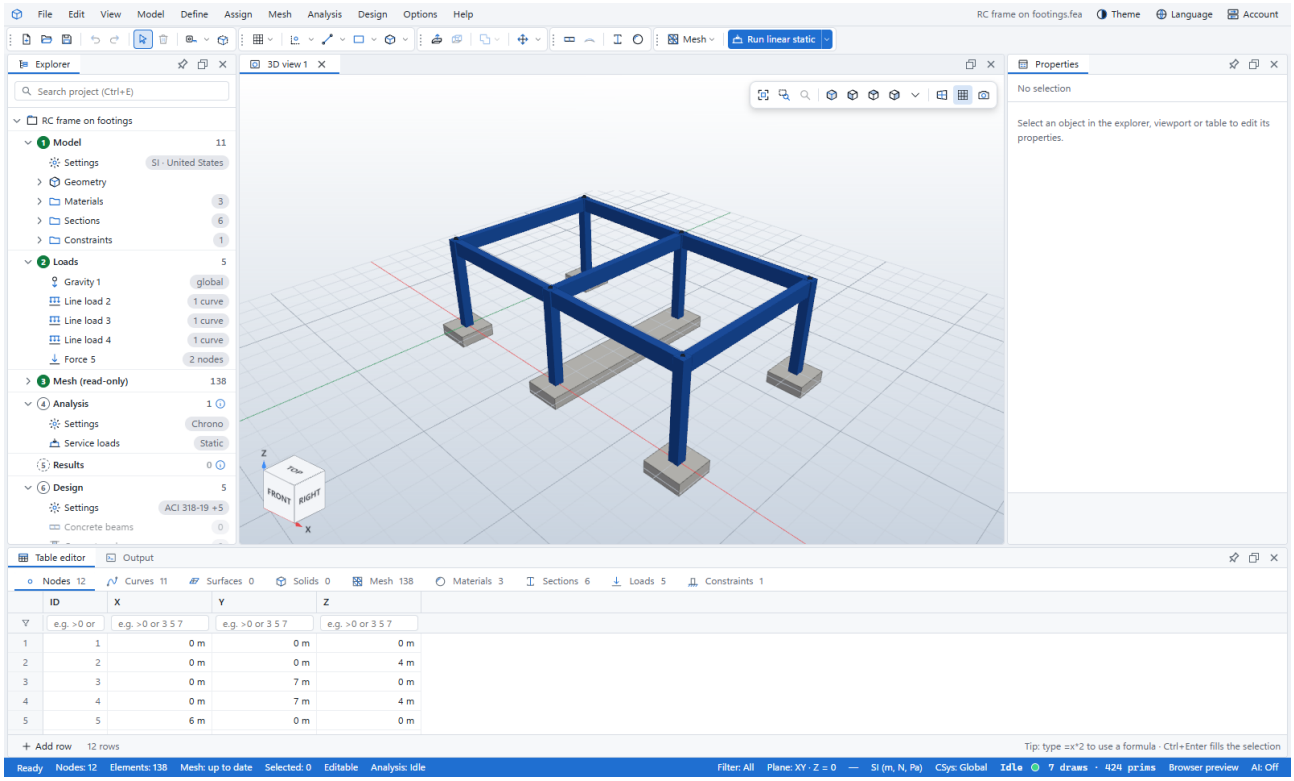


Figure 1. The FEA Studio window with a sample model open

The window has five areas:

Area	What it is for
Menu bar and toolbars	Every command. The toolbars hold the most common ones.
Explorer (left)	The project tree, arranged as the numbered stages of the work
3D view (centre)	The model. Several views and document tabs can be open at once.
Properties (right)	The settings of whatever is selected
Table editor and Output (bottom)	Spreadsheet editing of nodes, curves, materials and loads; the log of every action

The status bar at the bottom shows the mesh and analysis state, the selection filter, the work plane, the units and the coordinate system. Many of these items can be clicked. For example, clicking **Mesh:** rebuilds the mesh and clicking **Plane:** opens the work plane settings.

The Explorer stages

The Explorer arranges the project as numbered stages. Each stage has a status badge, so you can see at a glance what is done and what is still missing.

Stage	Contains	Typical badges
1 Model	Settings, Geometry (nodes, curves, surfaces, solids, groups), Materials, Sections, Constraints	empty · no supports · 11 objects · imported
2 Loads	Load cases and their loads	2 cases · 5 loads · none

Stage	Contains	Typical badges
3 Mesh (read-only)	Mesh settings, elements by type, mesh nodes, quality	not built · out of date · up to date
4 Analysis	Settings (the solver of each study), Studies	no studies · 1 study · not run · run
5 Results	Result sets	after analysis
6 Design	Settings (design settings, load combinations), Beam lines, column stacks, slabs, footings, connections…	items with ✓ or ✗ and a utilisation
7 Drawings	The drawing set	none yet · 12 drawings
8 Reports	Model, analysis and design reports	3 of 3 ready

Design, Drawings and Reports appear only when they have content, and the stages are renumbered when some are hidden. The **Settings** item of a stage opens the dialogs that set it up: **Model › Settings** opens **Project settings** (name, description, unit system), **Analysis › Settings** the solver of each study (Part 2), and **Design › Settings** the design settings and load combinations (Part 3).

- Use the search box at the top (`Ctrl+E` or `Ctrl+F`) to find any item by name.
- Press `F2` to rename the selected item.
- Right-click any folder or item for the commands that apply to it. For example, the Materials folder offers **Add material** and **Open in table**, and the Sections folder offers **Add profile** and **Add from section library…**.

Menus

The menu bar of FEA Studio has these menus: **File**, **Edit**, **View**, **Model**, **Define**, **Assign**, **Mesh**, **Analysis**, **Results**, **Design**, **Drawings**, **Options** and **Help**. At the right of the title bar are **Theme**, **Language** and **Account**.

- **Model** draws the geometry.
- **Define** sets up what the model is made of and loaded by: **Materials…**, **Steel sections…**, **Section designer**, **Load cases** and **Load combinations**.
- **Assign** gives the selected objects a material, section or shell, loads and supports, and runs the load generators.

Menus open as wide panels with commands grouped under headings. Each panel has a search box, and its shortcut keys are shown beside the commands.

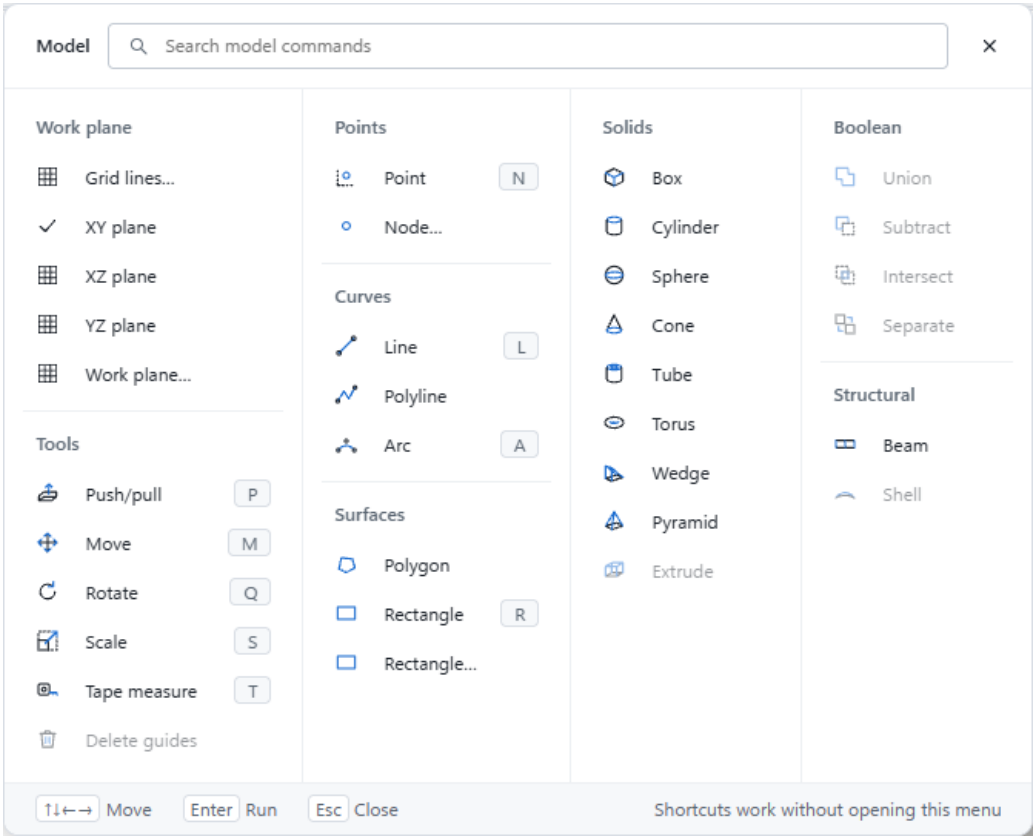


Figure 2. The Model menu: work plane, drawing tools, primitives, booleans and structural commands

- **Simplified menus** (Options › Workspace) is on by default. It shows only the menus the current stage needs:
  - While the model can be edited, you see **Model**, **Define**, **Assign** and **Mesh**.
  - Once the model has results, these are replaced by **Results** and **Drawings**.
- **Menu style** (Options › Workspace) switches between the modern panels and classic drop-down menus.
- Press `Alt` or `F10` to reach the menus from the keyboard.

Toolbars

The toolbars run below the menu bar. Press `Ctrl+F1` to show or hide them.

Toolbar	Commands
Standard	New, Open, Save · Undo, Redo · Select, Delete · Measure ▾ · Clipping box
Draw	Work plane ▾ · Points ▾ · Curves ▾ · Surfaces ▾ · Solids ▾
Modify	Push/pull, Extrude · Boolean ▾ · Transform ▾ (Move, Rotate, Scale)
Properties	Beam, Shell · Steel sections, Materials
Analysis	Mesh ▾ (Show mesh, Check quality) · **Run linear static** ▾ (the drop-down picks another analysis type)
Results	Shown after an analysis: result set, quantity, case, Display, scale, Design..., D/C ratios, Unlock model

Panels and layouts

You can drag panels to dock them anywhere, stack them as tabs, pin them or float them.

- **Options › Panels** shows or hides each panel: Explorer, Properties, Table editor, Output, Mesh quality, Connection and Template panel.
- **Options › Workspace › Layouts** switches between ready-made arrangements (**Modeling**, **Analysis**, **Viewport focus**). Use **Save current layout as…** to keep your own arrangement and **Reset layout** to go back to the default.
- Panels follow the stage. While you model, the Results panel is hidden. Once the model has results, the Table editor, Mesh quality and Template panels are hidden. They come back in their old place when you return to that stage.

Starting a project

The Start dialog

When FEA Studio opens with nothing loaded, it shows **Welcome to FEA Studio**. **File › New…** (`Ctrl+N`) and **File › Samples** open it again at the matching page.

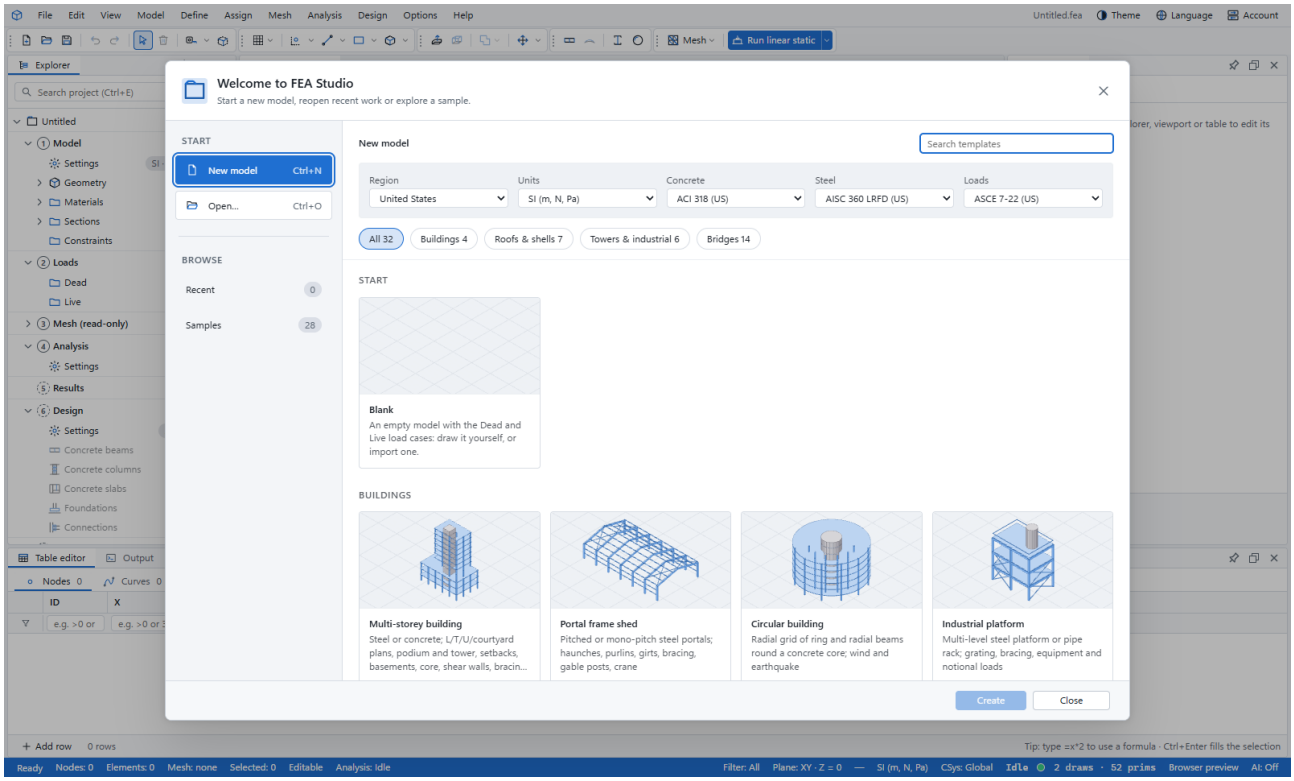


Figure 3. The Start dialog: New model with the region, units and design codes, and the template gallery

Option	What it does
New model (`Ctrl+N`)	Opens the template gallery with the units and design-code bar
Open… (`Ctrl+O`)	Opens a .fea project or an importable model file
Recent	Files you opened before (desktop app)
Samples	28 ready-made example models grouped by type

Samples open as new, unsaved models, so the original files are never changed. Use the chips (Simple, Complex, Concrete beams, Concrete slabs, Composite design, Shear walls, Foundations, Steel connections, IFC models) or the search box to find one.

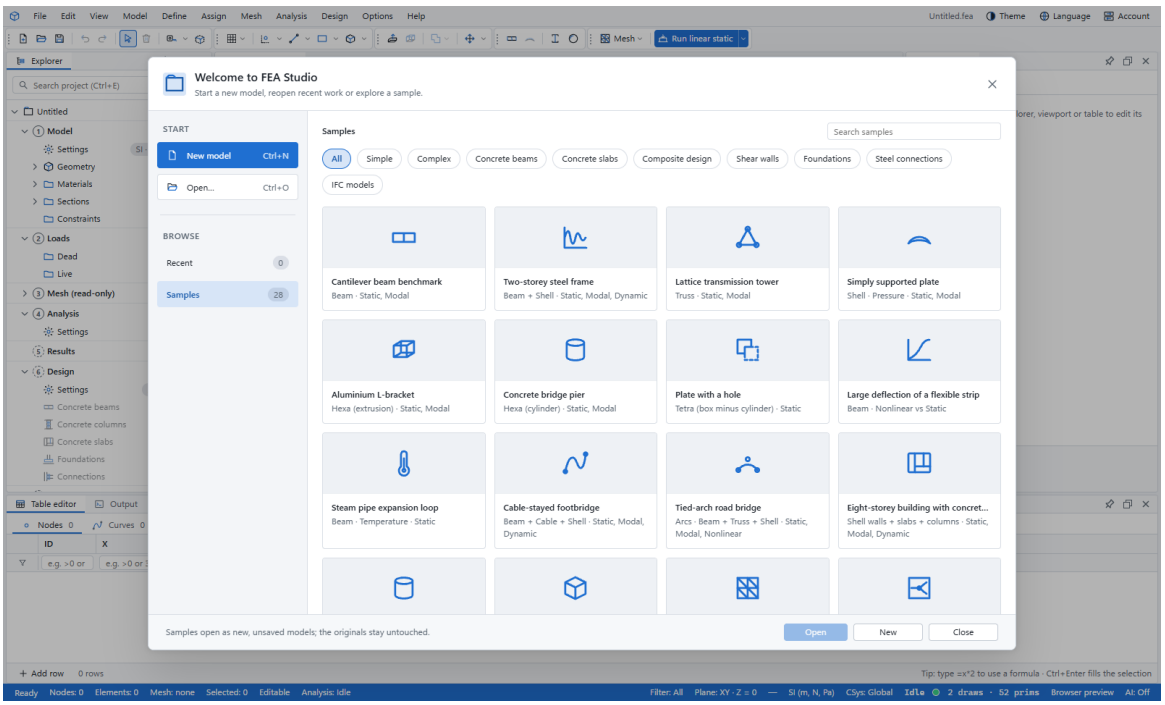


Figure 4. Samples in the Start dialog

New model: region, units and design codes

File › New... (`Ctrl+N`) opens the **New model** page. Pick the units and design codes first, then choose a blank model or a template.

Field	Choices
Region	United States, Korea, Europe, Canada, Australia, India, China, Philippines, Custom
Units	SI (m, N, Pa) · SI (mm, N, MPa) · US (in, lbf, psi)
Concrete	ACI 318 · KDS 14 20 · Eurocode 2 · CSA A23.3 · AS 3600 · IS 456 · GB 50010 · NSCP 2015
Steel	AISC 360 LRFD · AISC 360 ASD · KDS 14 31 · Eurocode 3 · CSA S16 · AS 4100 · IS 800 · NSCP 2015
Loads	ASCE 7-22 · KDS 41 · Eurocode

Choosing a **Region** fills in all the other fields:

Region	Units	Concrete	Steel	Loads
United States	US (in, lbf, psi)	ACI 318	AISC 360 LRFD	ASCE 7-22
Korea	SI (mm, N, MPa)	KDS 14 20	KDS 14 31	KDS 41
Europe	SI (m, N, Pa)	Eurocode 2	Eurocode 3	Eurocode
Canada	SI (mm, N, MPa)	CSA A23.3	CSA S16	ASCE 7-22
Australia	SI (mm, N, MPa)	AS 3600	AS 4100	ASCE 7-22
India	SI (mm, N, MPa)	IS 456	IS 800	ASCE 7-22
China	SI (mm, N, MPa)	GB 50010	Eurocode 3	Eurocode
Philippines	SI (mm, N, MPa)	NSCP 2015	NSCP 2015	ASCE 7-22

The region also sets the default drawing standard and sheet size; see Part 4. If you change one code by hand, the region becomes Custom, unless the new set of codes matches another region exactly.

Tip Pick the region before you choose a template. The codes you set here carry through to load combinations, member design, footings, connections and drawings, so you will not have to set them again in each module.

Templates

Below the setup bar is the template gallery. A template builds a complete parametric model: members, supports, load cases and studies. It then meshes the model and runs its load generators, with a progress card while it works.

The first card, **Blank**, gives an empty model with the load cases **Dead** (self-weight on) and **Live**. Use it to draw a model yourself or to import one.

Family	Templates
Buildings (4)	Multi-storey building · Portal frame shed · Circular building · Industrial platform
Roofs & shells (7)	Double-layer grid · Arch roof truss · Barrel vault · Dome · Hemispherical shell with a hole · Hyperbolic paraboloid · Cooling tower
Towers & industrial (6)	Lattice tower · Transmission tower · Chimney / stack · Storage tank / silo · Pipe rack · Stair
Bridges (14)	Bridge superstructure · Single-span overpass · Multi-span overpass · Waterway crossing · Straight flyover · S-curve flyover · U-turn flyover · 270° ramp · Truss bridge · Plate girder bridge · Box girder bridge · Tied arch bridge · Cable-stayed bridge · Suspension footbridge

To create a model:

1. Choose a family chip, or type in **Search templates**.
2. Click a card, then click **Create**. You can also double-click the card, or select it and press `Enter`.
3. The model is built, meshed and loaded. It opens ready for analysis.

Changing template parameters later

Options › Panels › Template panel shows the template's parameters, grouped as in the template form. Every change regenerates the model as one undo step.

- If you have edited the model by hand, FEA Studio asks **Regenerate the model?** first.
- The panel is blocked while the model has results; unlock the model first (see "Locked models").

Bridge components

In a bridge template you can change components one at a time. Click a girder, diaphragm, bearing, pier or abutment in the 3D view, then choose in **Changes apply to** how far the change should reach. For example:

- **This girder (G1, span 2)**
- **G1 in every span**
- **Every girder of span N**
- **Every exterior girder**
- **Every girder**

Component	What you can change
Girder	Section (for example AASHTO BT-63), end blocks, plate dimensions
Diaphragm	Present, Thickness, Depth
Bearing	Type (From the layout, Pot fixed, Pot guided (free along / across), Pot free, Elastomeric), plan size, height
Pier	Single column, Multiple column, Wall column, Multiple pile, Box girder (flared column); foundation type (isolated or combined footing, pier pile, pile shaft, pile group)
Abutment	This abutment or both

Use **Select them** to see every component a change affects, **Copy the changes of this … to …** to repeat a change elsewhere, and **Reset to template** or **Reset all** to undo your changes.

Opening, importing and saving

Command	What it does
File › Open (`Ctrl+O`)	Opens a .fea project
File › Recent	Recently used files (desktop app)
File › Samples	The sample models
File › Save (`Ctrl+S`) / Save as… (`Ctrl+Shift+S`)	Writes the .fea project file. In the browser the file is downloaded.
File › Import	Abaqus input (.inp), Abaqus/CAE (.cae), Abaqus output database mesh (.odb), Nastran bulk data (.bdf, .dat, .nas), MIDAS Civil / Gen (.mct, .mgt, .mcb, .mgb, .mgbx), ETABS text model (.e2k, .$et), nodes from CSV
File › Import IFC	Builds a structural model from an IFC building model (Revit, ArchiCAD, Tekla and others); .ifczip is accepted
File › Import Revit model	Reads a Revit .rvt file directly and builds the structural model from it
File › Update from newer IFC · Load IFC model	Brings changes from a newer IFC of the same building; shows an IFC model in the background
File › Export	Abaqus .inp, MIDAS .mct, Nastran .bdf, VTK, nodes and elements as CSV, IFC (structural analysis model, physical model, or with results), a PNG image of the 3D view; **Export connections to IFC…** writes the steel connections

You can also drop a file anywhere on the window to open it.

If you close FEA Studio with unsaved changes, it asks whether to **Save**, **Don't save** or **Cancel**. After a crash, it offers to **Recover** the last state.

Import IFC

Import IFC opens a dialog that lists the element types found in the file. Tick the roles you want: columns, beams, braces, slabs, walls, footings, bearings and other elements. The other settings are:

Setting	Default	Meaning
Joining tolerance	300 mm	Ends closer than this are joined
Drop faces narrower than	150 mm	Removes slivers

Setting	Default	Meaning
Joints	Rigid links (members stay on their axes)	Or **Move ends onto the axes (centre lines)**
Supports at the base	Fixed	Or Pinned
Walls	Load-bearing (or concrete / masonry)	Or All walls, or None
Slabs on grade	Off	Include ground slabs
Sloped slabs	Flat plate, average depth	Or thinnest part, thickness below, or leave out
Area loads	Superimposed dead 1.5 kPa · Live (floors) 3 kPa · Live (roofs) 1 kPa	Added as load cases
After import	Make design objects · Show the BIM model	Beam lines, stacks and so on; the IFC shown in the background

If the IFC file also contains a structural analysis model, FEA Studio asks whether to use the **Building elements** or the **Analysis model**.

Project information

Click the project name at the top of the Explorer to see the project sheet in Properties, or double-click **Model › Settings** in the Explorer for the **Project settings** dialog.

- **Name** and **Description** appear on every report cover and page header.
- **Unit system** sets the units used for input and in reports. Values are always stored in SI.

You can also change the units from the units item in the status bar.

Navigating the 3D view

Mouse and view cube

- **Left drag** orbits the view, **right drag** or **middle drag** pans, and the **mouse wheel** zooms at the cursor.
- **Click the view cube** on a face, edge or corner to turn to that view. Drag the cube to orbit.
- **Double-click** zooms to the selection.
- With no tool active, the **arrow keys** orbit in 15° steps, **Shift+arrows** pan, and **Page Up / Page Down** zoom.

Other mouse layouts are available under **Options › Workspace › Settings… › Mouse navigation**:

- FEA (left drag orbits), the default
- Revit, AutoCAD, Navisworks (middle button)
- Tekla Structures (Ctrl+middle orbits)
- Abaqus/CAE (Ctrl+Alt)

In every layout except FEA, a left drag draws a selection box.

View commands

Command	Key	What it does
Fit all	`F`	Fits the whole model
Zoom to selection	`Z`	Fits the selected objects
Zoom window	`W`	Drag a box to zoom into
Normal to selection	`V`	Looks straight at the selected face
Home view / Set as home view	`H`	Returns to, or stores, your favourite view
Walk	`Shift+W`	Drag to look; W A S D to move, Q / E down and up, Shift faster, Esc to end
Isometric / Front / Top / Right	`0` / `1` / `7` / `3`	Standard views
Orthographic	`5`	Switches between perspective and orthographic
Isolate, Hide, Show all	–	Shows only the selection, hides it, or brings everything back

Clipping box

View › Clipping › Clipping box (also on the Standard toolbar) puts a box around the model.

- Drag its coloured arrows to cut the model along X (red), Y (green) or Z (blue). Solid faces that are cut are drawn filled.
- **Clipping arrows** hides the handles but keeps the cut.
- **Reset clipping box** removes the cut.

Selecting

- **Click** to select an object, and **Ctrl+click** to add or remove one.
- **Shift+drag** draws a selection box. Dragging left to right selects what is fully inside; dragging right to left also selects what the box crosses.
- **Esc** clears the selection.
- `Ctrl+A` selects all and `Ctrl+I` inverts the selection.
- **Edit › Selection › Filter** (or **Filter:** in the status bar) limits picking to one kind of object, for example **Nodes only** or **Curves only**.

Right-click menu

Right-clicking in the 3D view gives the commands that fit the current selection:

Selection	Commands offered
Nothing	Paste, Select all, Run analysis, Show mesh, Fit all, standard views, Grid, Loads, clipping, Show all, Filter
2 nodes	Line between nodes, Beam between nodes
3 nodes	Arc through nodes, Surface from nodes
4 or more nodes	Polyline through nodes, Surface from nodes
Curves	Assign section…
Surfaces	Assign shell…, Extrude…
2 or more solids	Boolean ▸
Anything	Apply load ▸, Add support ▸, Mesh ▸, Cut, Copy, Paste, Duplicate, Move, Rotate, Scale, Flip along ▸, Make group, Delete, Isolate, Hide, Properties

Drawing the geometry

The geometry is made of **nodes**, **curves** (lines, polylines and arcs), **surfaces** and **solids**. Curves become beams, trusses or cables; surfaces become shells; solids become solid elements. You give each one a section or material in a later step (see "Materials and sections").

Structural grid

A grid of named axes and levels makes drawing quick, because the drawing tools snap to every grid crossing at every level.

Model › Work plane › Grid lines…

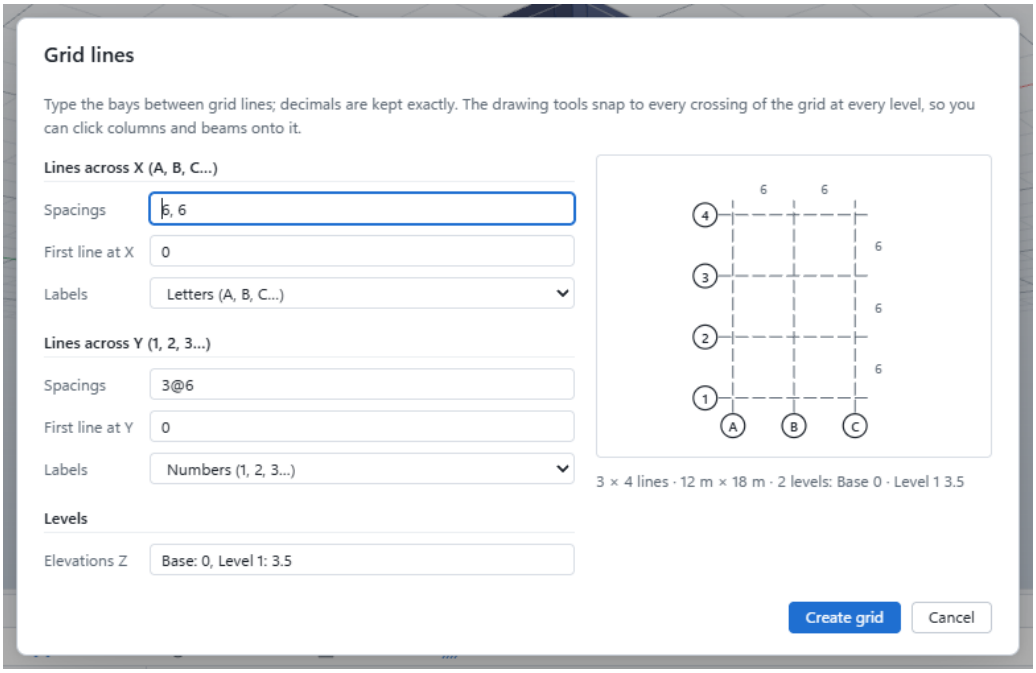


Figure 5. The Grid lines dialog with a live preview

Field	Example	Notes
Lines across X (A, B, C…) · Spacings	`6, 6` or `5@6.125`	Bay widths; repeats, units (`6125 mm`) and formulas (`=36.7/2`) are allowed
First line at X	`0`	Where the first axis lies
Labels	Letters (A, B, C…) or Numbers (1, 2, 3…)	
Lines across Y (1, 2, 3…) · Spacings	`3@6`	
Levels · Elevations Z	`Base: 0, Level 1: 3.5`	Name: elevation, separated by commas

Click **Create grid**. Later, the same command opens **Edit grid lines**, where you can use **Update grid** or **Remove grid**. **Options › Display › Grid lines** shows or hides the grid bubbles and level tags.

Work plane

The drawing tools draw on the work plane unless they snap to existing geometry.

- **XY plane** draws on a horizontal plane (plans, floors). It passes through the selected node, or else lies at Z = 0.

- **XZ plane** and **YZ plane** are vertical planes for elevations.
- **Work plane…** sets the plane and its position (**At**) exactly, and can turn the view to look straight at it.

The current plane is shown in the status bar, for example **Plane: XY · Z = 0**.

Points and curves

Tool	Key	How to use it
Point	N	Click to place nodes. They snap to endpoints, edges, faces and the grid.
Node…	–	Type X, Y, Z in a dialog
Line	L	Click the start, then the end, or type a length. Clicks chain lines together; Enter ends the chain. With two nodes selected, the line is made at once.
Polyline	–	Click points; Enter or a double-click finishes
Arc	A	Start point, a point on the arc, end point; or select 3 nodes first

Surfaces

Tool	Key	How to use it
Polygon	–	Click the corners, then close it by clicking the first point, pressing Enter or double-clicking
Rectangle	R	First corner, then the opposite corner, or type `length, width`. It lies on the face you start on, or else on the work plane.
Rectangle…	–	Dialog with Corner X/Y/Z, Width (X) and Depth (Y)

Typing exact values

While a tool is active, start typing to open the **Measurements** box at the cursor. Press Enter to apply the value.

You type	Meaning
`4` or `2500 mm`	A length along the current direction
`3, 2`	Absolute x, y on the work plane (or a rectangle's length, width)
`@1.5, 2`	Relative offset dx, dy
`@4<30`	Relative length 4 at 30°
`3*cos(rad(30))`	A formula

Inference snapping

As you move the cursor, FEA Studio shows what it snaps to: **Endpoint**, **Midpoint**, **On Edge**, **On Face**, **Intersection**, **Grid Intersection**, **Guide Point**, **On Guide**, **On Red/Green/Blue Axis**, **From Point** and **Parallel to Edge**.

- Press → to lock the red X axis, ← for the green Y axis and ↑ for the blue Z axis. Press ↓ to release the lock.
- Hold Shift to keep the current snap while you move.

- `Backspace` removes the last point, `Enter` finishes and `Esc` cancels.
- While a tool is active, the right-click menu offers **Finish**, **Undo last point**, **Cancel** and the axis locks.

Solids

Model › Solids adds primitive solids through a dialog. Each dialog takes an origin (X, Y, Z), an axis (X, Y or Z) where it applies, the dimensions and a material. You can change the dimensions and rotation later in Properties.

Primitive	Origin	Dimensions (defaults)
Box	Corner	Size X, Y, Z (1)
Cylinder	Base centre	Radius 0.5, Height 2
Sphere	Centre	Radius 0.5
Cone	Base centre	Base radius 0.5, Top radius 0 (more than 0 gives a frustum), Height 1
Tube	Base centre	Outer radius 0.5, Inner radius 0.4, Height 2
Torus	Centre	Ring radius 1, Tube radius 0.25
Wedge	Corner	Length X 2, Width Y 1, Height Z 1
Pyramid	Base centre	Base width 1, Base depth 1, Height 1

Extrude turns selected surfaces into solids along a vector you type.

Boolean operations

Select two or more solids, then choose a command from **Model › Boolean**:

- **Union** joins them.
- **Subtract** cuts the other solids away from the first one you selected.
- **Intersect** keeps the common part.
- **Separate** undoes a boolean.

The original solids stay editable as parts of the result.

Push/Pull, Move, Rotate and Scale

Tool	Key	How to use it
Push/pull	`P`	Click a surface or solid face, move the mouse and click, or type a distance. Clicking a line sweeps it into a surface, and ↑ pulls it straight up. A double-click repeats the last distance.
Move	`M`	Base point, then the destination, or type a distance
Rotate	`Q`	Centre (a protractor lies on the face under the cursor; the arrow keys pick the axis), then the start and end of the angle, or type an angle
Scale	`S`	Fixed point, reference point, then a factor
Tape measure	`T`	Measures between two points and leaves guide lines or guide points. Click on an edge to make a parallel guide. **Delete guides** removes them all.

> **Tip** To make copies with Move or Rotate, tap `Ctrl` to switch copy mode on. Right after the first copy, type `x5` to make 5 copies at the same spacing, or `/5` to spread 5 copies over the distance.

The same transforms are available as dialogs from **Edit › Transform**:

- **Move…** takes ΔX, ΔY, ΔZ.
- **Rotate…** takes the axis, angle and centre.
- **Duplicate…** (`Ctrl+D`) takes an offset and the number of copies. It also has a **Copy supports and loads** option and shows a live preview.

Edit › Flip mirrors the selection about its centre along X, Y or Z.

Groups, clipboard and undo

- **Make group** (`Ctrl+G`) groups curves, surfaces and solids. Clicking one part selects the whole group; `Alt+click` selects a single part. **Ungroup** is `Ctrl+Shift+G`.
- **Cut, Copy, Paste** work as usual. Pasted nodes that land on existing nodes are merged with them.
- **Undo** is `Ctrl+Z` and **Redo** is `Ctrl+Y`. Every edit can be undone, including template regeneration.

The Table editor

The **Table editor** at the bottom lists Nodes, Curves, Surfaces, Solids, Mesh, Materials, Loads and Constraints in tabs.

- You can edit cells directly and filter columns (for example `>0`).
- Type a formula such as `=x*2` in a cell.
- `Ctrl+Enter` fills the selected cells with the same value.

Materials and sections

The Define menu

The **Define** menu sets up what the model is made of and how it is loaded: materials, sections, load cases and load combinations.

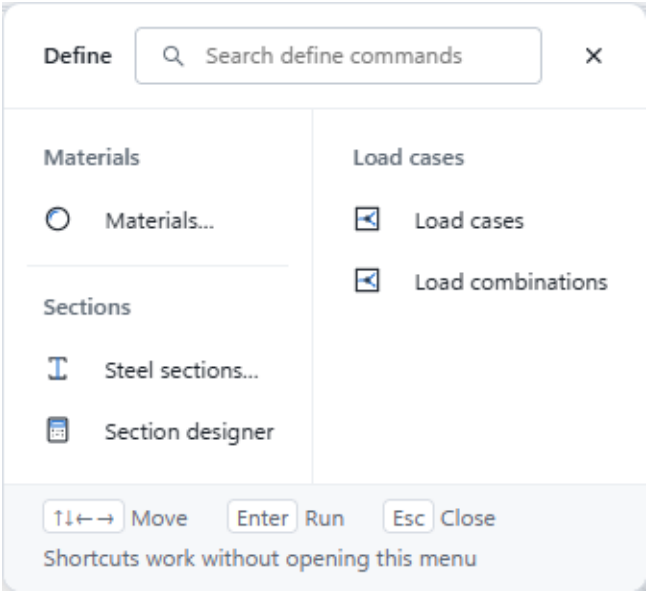


Figure 6. The Define menu

Materials

Define › Materials... opens the **Materials** dialog. It lists the project's materials on the left; select one to edit it on the right, and click **Apply** to keep your changes.

- **Add material** creates a new material. Choose its **Type**: Concrete, Steel, Rebar or Other. The type decides which strengths it has: f′c for concrete, fy and fu for steel and rebar.
- **Add from the library** picks materials from the built-in libraries, and from your own library of materials you saved before. Tick the sources and the materials you want. The built-in libraries include:
  - Structural steel (EN 10025): S235, S275, S355, S460
  - Steel (ASTM): A36, A572 Gr. 50, A992, Stainless 304, Stainless 316
  - Concrete (EN 1992): C20/25 to C50/60
  - Aluminium and others: Al 6061-T6, Al 6082-T6, Ti-6Al-4V, Timber C24, Glulam GL24h
- Every material has **Elastic properties** (Young's modulus E, Poisson's ratio ν) and **Physical properties** (density ρ, thermal expansion α).

The concrete and steel design modules (Part 3) take f′c and fy from the material of each member. You can also select a material in the Explorer's Materials folder and edit it in Properties.

Sections (profiles)

Define › Steel sections... opens the **Section library**.

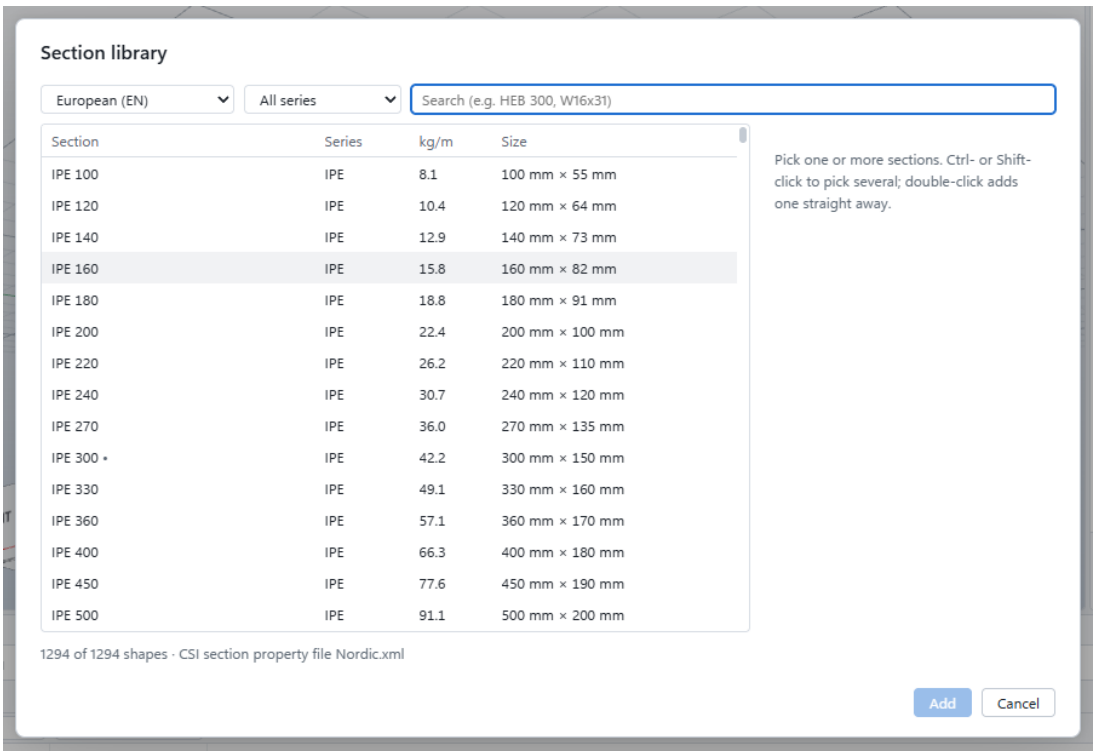


Figure 7. The Section library: catalog, series filter and search

1. Pick a catalog: AISC (US), CISC (Canada), European (EN), UK, KS (Korea), AS/NZS or IS (India).
2. Narrow the list with the series filter, or type a name such as `HEB 300` or `W16x31`.
3. Select one or more sections (`Ctrl+click` or `Shift+click`) and click **Add**. A double-click adds one section at once.

To make your own section, right-click the Sections folder and choose **Add profile**. The shapes are Rectangle, Circle, Pipe, Box, I-Section, Channel, Tee, Angle and General. Enter the dimensions in the profile's Properties afterwards. Properties also shows the computed Area, Iy, Iz, J, the extreme fibres and the principal axes.

Define › Section designer opens the section designer as a tab of the main area: draw a section of any shape, read its properties, and click **Use in model** to add it to the model's sections. The designed sections are saved in the project.

Assigning sections to curves

1. Select the curves.
2. Choose **Assign › Properties › Section...** (or **Model › Structural › Beam**, or right-click › **Assign section...**).
3. Choose the **Element** type (Beam, Truss or Cable), the **Material** and the **Profile**.

With no curves selected, **Beam** adds a new beam between two nodes that you choose in the dialog. **Section...** also offers **New pipe** for a tube of any size (a monopile or a tower leg). **Assign › Properties › Material...** changes only the material of the selected curves, shell surfaces and solids.

The curve's Properties then control how the section sits:

- **Section rotation (°)** turns the section about the member axis. At 0°, local z points to global Z (for vertical members, to global X).

- **Section position** places the section relative to the curve: Default, Centroid, Top of section, Bottom of section or Custom. With Default, level beams hang below the curve with their top flush, while columns and braces stay centred.
- **Offset y / Offset z** move the section away from the nodes. It stays rigidly tied to them.

Assigning shells to surfaces

1. Select the surfaces.
2. Choose **Assign › Properties › Shell** (or **Model › Structural › Shell**, or right-click › **Assign shell…**).
3. Enter the **Thickness** and the **Material**.

In the surface's Properties, **Section position** (Default, Mid-surface, Top face, Bottom face or Custom) and **Offset** set where the shell sits.

> **Note** Turn on **Options › Display › Realistic** to see members and shells drawn with their true sections and thicknesses. **Curve lines** adds the centre lines on top.

Supports

Select nodes, curves, surfaces or solid faces, then choose a command from **Assign › Constraints**:

Command	Restrains
Fixed	All six degrees of freedom
Pin	The three translations; rotations stay free
Roller	The vertical translation (Uz) only
Symmetry	A symmetry plane; choose the plane normal X, Y or Z
Custom	Each of Ux, Uy, Uz, Rx, Ry, Rz set to Restrained or Free

A support on a curve, surface or face applies to every mesh node on it. The Explorer's Model stage shows **no supports** until at least one exists, as a reminder before you analyse.

Loads

Load cases

Each load belongs to a load case, and each case has a category. The category tells the design codes how to combine the case with the others.

Define › Load cases › Load cases

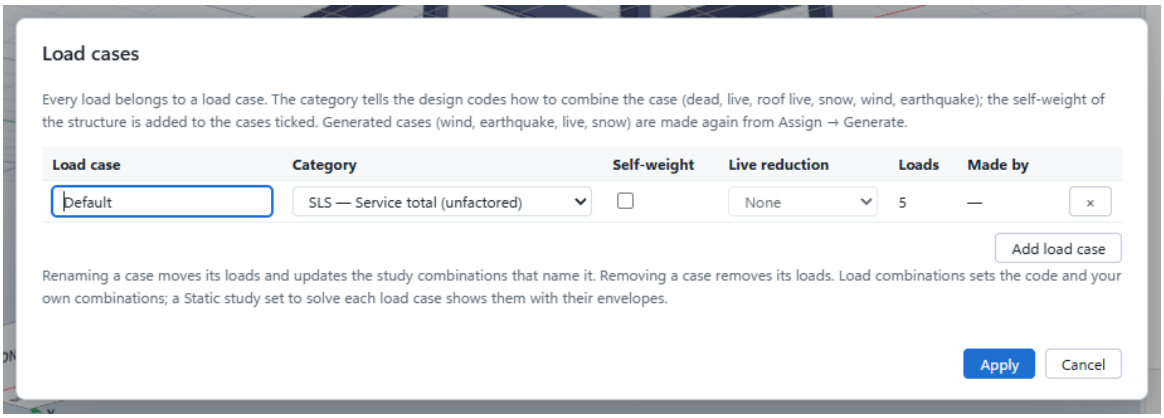


Figure 8. The Load cases dialog

Column	Meaning
Load case	The name, for example `Dead`, `Live` or `Wind X`
Category	Dead (permanent) · Live (imposed) · Roof live · Snow · Wind · Earthquake · Notional (steel stability) · Service total (unfactored) · Factored total
Self-weight	Adds the self-weight of the model to this case
Live reduction	Live-load reduction by tributary area
Loads / Made by	How many loads the case holds, and whether you or a generator made them

Click **Add load case** to add a case, then **Apply**. Renaming a case moves its loads with it; removing a case removes its loads.

Applying loads

Select the objects first, then choose a command from **Assign › Loads**. Each load dialog has a **Load case** field; typing a new name there creates a new case.

Load	Applies to	Fields (defaults)
Force	Nodes, curves, surfaces	Magnitude (1000 N), Direction X/Y/Z (0, 0, −1)
Moment	Nodes, curves, surfaces	Magnitude (1000 N·m), Axis (0, 0, 1)
Line load	Curves	Load per length (10 kN/m), Direction, **Per horizontal metre (projected)**
Area load	Surfaces	Load per area (2 kPa), Direction, **Per plan area (projected)**, Load panel distribution
Pressure	Surfaces, solid faces	Pressure, normal to the face
Temperature	Nodes, curves, surfaces, solids	Temperature change (°C)

Load	Applies to	Fields (defaults)
Gravity	The whole model	Acceleration (9.81 m/s²)

- A force on a curve or surface is a total, shared equally among its mesh nodes.
- **Load panel distribution** (One-way or Two-way) lets a floor without shells pass its area load to the edge beams.
- **Options › Display › Loads** shows or hides the load arrows.

Load generators

Assign › Generate makes code loads for you. Each generator first asks for the **Code**, then for its inputs. It remembers the inputs of the last run.

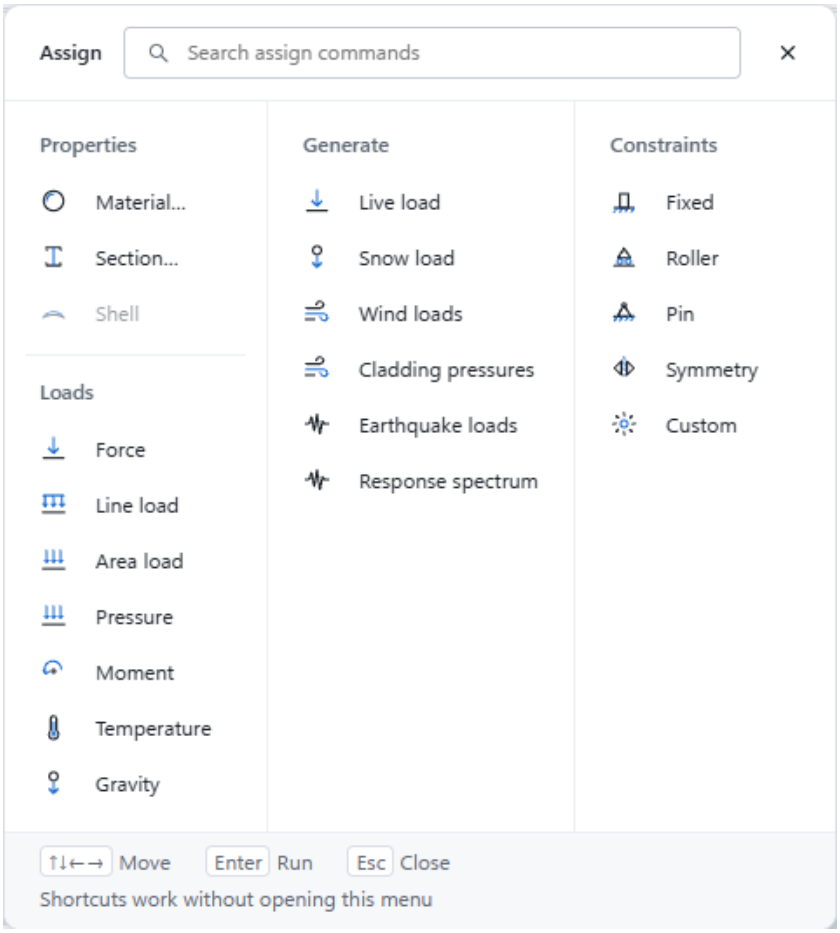


Figure 9. The Assign menu: properties, loads, load generators and supports

Generator	Codes	Main inputs
Live load	ASCE/SEI 7-22 Table 4.3-1 · KDS 41 10 15 · EN 1991-1-1 Table 6.2	Occupancy (gives the load from the code table), load case, load panel distribution, live load reduction by tributary area
Snow load	ASCE/SEI 7-22 Ch. 7 · EN 1991-1-3 · KDS 41 12 00	Ground snow, Ce, Ct, Is, slippery roof; balanced, unbalanced, drift at a roof step, or sliding from an upper roof
Wind loads	ASCE/SEI 7-22 Ch. 27 · EN 1991-1-4 · KDS 41 12 00	Basic wind speed, exposure or terrain category, factors (Kd, Kzt, Ke, G or Iw, GD), directions, roof uplift, enclosure, width across the wind
Cladding pressures	As for wind	Effective wind area, roof pitch; shows a table of zone pressures

Generator	Codes	Main inputs
Earthquake loads	ASCE/SEI 7-22 §12.8 · KDS 41 17 00 · EN 1998-1 §4.3.3.2	SDS, SD1, TL, R, Ie (or agR, ground type, q for EC8); lateral system; accidental eccentricity; live-load share of the seismic weight. Makes the cases **EQ X** and **EQ Y**.
Response spectrum	As for earthquake	Makes a response spectrum study (see Part 2)

Tip Select the surfaces first for live and snow loads. Wind and earthquake loads use the whole model.

Load combinations

Define › Load cases › Load combinations shows the combinations the design code builds from your load-case categories, and lets you add your own. Part 3 explains this dialog in full, because the combinations drive the design.

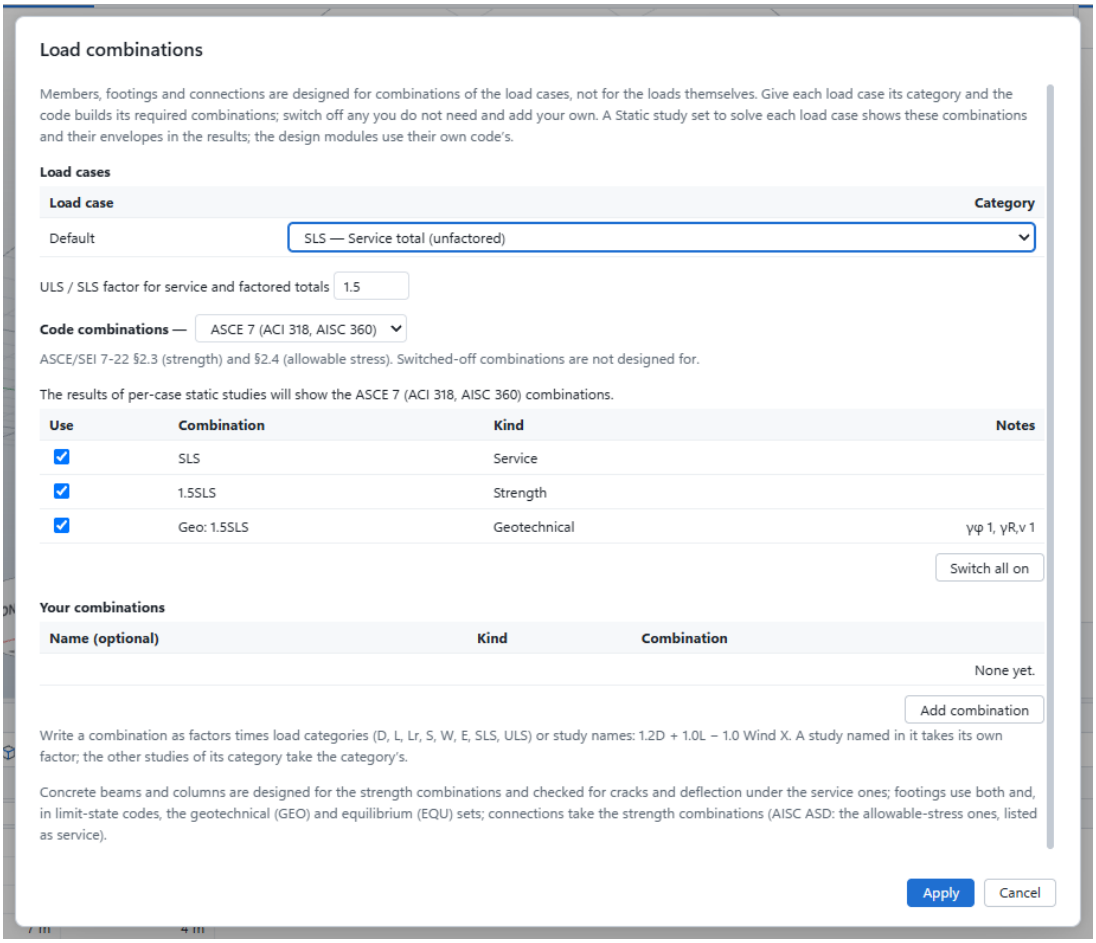


Figure 10. The Load combinations dialog

Meshing

FEA Studio analyses a mesh made from your model. You do not have to build it yourself: the mesh is generated or updated automatically when you run an analysis. Still, you can control it and inspect it.

Mesh settings

Mesh › Generate › Settings

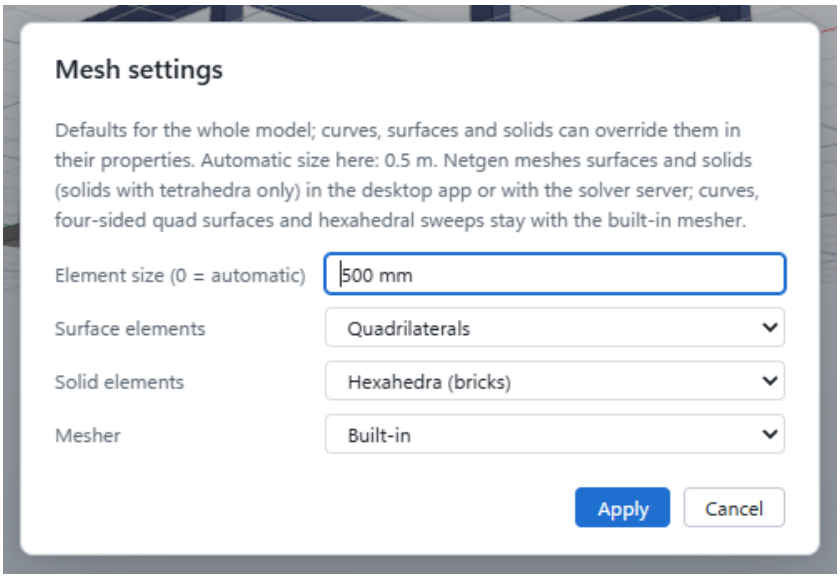


Figure 11. Mesh settings

Setting	Choices
Element size (0 = automatic)	A length; the note shows the automatic size
Surface elements	Quadrilaterals or Triangles
Solid elements	Hexahedra (bricks) or Tetrahedra
Mesher	Built-in or Netgen

Curves, surfaces and solids can override these settings in their Properties, with their own element size, divisions or element shape. **Mesh › Divisions…** sets the number of elements per segment for the selected curves.

> **Note** The Netgen mesher works in the desktop app, or in the browser when the solver server is running. It meshes solids with tetrahedra only.

Mesh commands

Command	Key	What it does
Generate mesh	`Ctrl+M`	Builds the mesh now (`Esc` cancels)
Refine mesh / Coarsen mesh	–	Halves or doubles the element size of the selection, or of the whole model
Clear mesh	–	Removes the mesh
Show mesh	–	Shows the mesh over the model (read-only)

Command	Key	What it does
Shrink, Mesh nodes	–	Display aids for checking the mesh

The status bar shows **Mesh: none**, **out of date** or **up to date**. Click it to regenerate the mesh.

Mesh quality

Mesh › Inspect › Check quality opens the **Mesh quality** panel.

- **Metric** chooses the measure: Scaled Jacobian, Aspect ratio, Minimum angle, Maximum angle or Warpage.
- **Grade chips** show the counts of Good, Fair, Poor and Invalid elements. Click a chip to select those elements.
- A histogram, the minimum, mean and maximum, and a list of the worst elements are shown below.
- **Quality map** colours the elements in the 3D view by quality.

Locked models

Once a study has been solved, the model is **locked** so the results always match the geometry. You will see **locked** next to the project name and **Locked** in the status bar, and editing commands are greyed out.

To edit the model again, choose **Results › Lock › Unlock model**, or click **Unlock model** on the Results toolbar. Unlocking removes all result sets; analyse again after editing. Design objects such as beam lines, slabs and footings stay, but their checks become out of date.

Settings, shortcuts and help

Settings

Options › Workspace › Settings... contains:

- **Theme**: Use system setting, Light or Dark.
- **Language**: applies at the next start.
- **Mouse navigation**: the mouse layouts described in "Navigating the 3D view".
- **Precise depth**.
- **Update feed URL** and **Check for updates at startup** (desktop app).

Keyboard shortcuts

Help › Learn › Keyboard shortcuts lists every shortcut. The main ones are:

Area	Shortcuts
File	`Ctrl+N` New · `Ctrl+O` Open · `Ctrl+S` Save · `Ctrl+Shift+S` Save as · `Ctrl+P` Report
Edit	`Ctrl+Z` / `Ctrl+Y` Undo / Redo · `Ctrl+X` `Ctrl+C` `Ctrl+V` · `Ctrl+D` Duplicate · `Delete` · `Ctrl+A` Select all · `Ctrl+I` Invert · `Alt+Enter` Properties
Drawing tools	`N` Point · `L` Line · `A` Arc · `R` Rectangle · `P` Push/pull · `M` Move · `Q` Rotate · `S` Scale · `T` Tape measure · `Ctrl+G` / `Ctrl+Shift+G` Group / Ungroup
Mesh and analysis	`Ctrl+M` Generate mesh · `F5` Run analysis · `Shift+F5` Cancel
View	`F` Fit all · `Z` Zoom to selection · `W` Zoom window · `V` Normal to · `H` Home · `Shift+W` Walk · `0` `1` `7` `3` Views · `5` Orthographic · `G` Grid
Workspace	`F1` Workflows · `Alt` or `F10` Menus · `Ctrl+E` Explorer search · `Ctrl+F1` Toolbars · `F6` Next pane · `Ctrl+Tab` Next tab · `Ctrl+F4` Close tab · `Shift+F10` Context menu · `F2` Rename

Guided workflows

File › Workflows (`F1`) opens step-by-step guides with buttons for each step. Guides are available for concrete beams, concrete slabs and foundations.

AI assistants

Help › AI › AI assistants (MCP)... connects FEA Studio to an AI assistant that can read and edit the model through FEA Studio's tools. This works in the desktop app only.

Desktop and browser

FEA Studio runs as a Windows desktop app and as a browser preview. Modeling works the same in both, with these differences:

Feature	Desktop app	Browser
Recent files	Yes	Not available
Save and export	Windows save dialog	The file is downloaded
Netgen mesher	Yes	Only with the solver server
Analysis	Built in	Needs the solver server
AI assistants (MCP)	Yes	Not available

Checklist before analysis

Before you go on to Part 2, check that:

1. Every member has a section and every shell a thickness and material. **Realistic** view makes gaps easy to spot.
2. The model has supports. The Model stage must not say **no supports**.
3. Every load is in a load case, and every case has the right category.
4. The mesh is **up to date**, or you are happy for it to be built when you run.
5. The project is saved (`Ctrl+S`).