ILX **Structures**

3D STRUCTURAL ANALYSIS AND DESIGN IN YOUR BROWSER

User Manual

Modelling, loads, analysis, design, reports and settings, step by step, for ILX Structures v0.1.0.

ILX Studio, LLC · services@ilxstudio.com

Version 0.1.0 · November 2026

ILX Structures User Manual

Version 0.1.0, November 2026. Covers the ILX Structures web application: 3D structural analysis, and design to AISC 360-16, AISC 341-22, ACI 318-19, NDS 2018 and TMS 402-16 with ASCE 7-22 loads.

Responsibility

ILX Structures helps you through the analysis and design and documents every check, but the engineer of record reviews and approves the results. See the full disclaimer at https://ilxstructures.com/legal/disclaimer.

Notices

ILX Studio, LLC is not affiliated with or endorsed by AISC, ACI, AWC, TMS, ASCE or any other code organization. ILX Structures works in your browser; nothing to install. Models are saved and analysed on ILX Studio's servers.

The latest version of this manual is always online at https://ilxstructures.com/manual. Questions and feedback: services@ilxstudio.com.

© 2026 ILX Studio, LLC. All rights reserved.

Contents

1 **User manual** 5

2 **Getting started** 6

Quick start 6
Quickstart wizard 6
Where things are 7

3 **The editor** 8

Ribbon tabs 8
Command palette 9
Saving, versions and conflicts 9
Jobs 9

4 **Modelling** 10

Nodes 10
Members 10
Plates 10
Supports and springs 10
Grid lines and stories 10
Groups and selection 11
Advanced model features 11
Import and export 11

5 **Loads and combinations** 12

Load cases 12
Applying loads 12
Environmental load generators 12
Combinations 12

6 **Analysis** 13

Run (F5) 13
Analysis types 13
Second-order effects 14
Stale results 14

7 **Results** 15

In the 3D view 15
Tables 15
Signs 15

8 **Design** 16
Steel design (AISC 360, LRFD) 16
Composite beam (AISC 360 Ch. I) 17
Concrete (ACI 318-19) 17
Wood (NDS) 17
Masonry (TMS 402-16) 17
Seismic design (AISC 341-22) 18
Story drift and wind drift 18
Direct analysis method helper 18

9 **Reports** 19
Content 19
Project information and title block 19
Output 19

10 **Bridge to ILX Connection and ILX Calcs** 20
Send to ILX Connection / ILX Calcs 20
Connection type per member end 20
Project navigator 20

11 **Units and sign conventions** 21
Units 21
Axes 21
Member end forces 22
Diagrams, sagging and hogging 22
Reactions and loads 22
Plate and solid results 22
Design ratios 22
Codes 22

12 **Settings and account** 23
Settings 23
Account 23
Support 23

13 **Keyboard and mouse** 24
Keyboard 24
Mouse 25

14 **Troubleshooting** 26

CHAPTER 1

User manual

The chapters of the manual, in order. Search the manual with the box on the left, or read it all as one PDF.

1. Getting started
2. The editor
3. Modelling
4. Loads and combinations
5. Analysis
6. Results
7. Design
8. Reports
9. Bridge to ILX Connection and ILX Calcs
10. Units and sign conventions
11. Settings and account
12. Keyboard and mouse
13. Troubleshooting

CHAPTER 2

Getting started

ILX Structures (version 0.1.0) is a 3D frame and plate finite-element program with steel, concrete, wood and masonry design, in the browser. Your projects and models are stored in your ILX account on the server; every save is a version you can restore. One plan includes every feature.

Quick start

1. **Create a project.** On the Projects page press **New project** and give it a name. A project holds models, the project information (job number, client, address) and the project's settings.
2. **Create a model.** In the project press **New model** and choose:
   - **Blank** (optionally with the standard load cases D, L, Lr, S, W, E),
   - **Template** (portal frame, space frame, planar truss, node grid) with its parameters,
   - **Worked example** (continuous beam, portal frame, simple truss),
   - **Import file** (`.ilxs`, DXF, IFC, STEP, IGES, ETABS `.e2k`, mesh files), or
   - **Quickstart wizard** (below).
3. **Model** the structure: nodes, members, plates, supports, sections and materials (Modelling).
4. **Load** it: load cases, applied loads and the ASCE 7-22 combinations (Loads).
5. **Analyze**: press **F5** to solve every combination with the envelope (Analysis).
6. **Look at the results**: deformed shape, force diagrams, reactions, tables (Results).
7. **Design**: Design ▸ Setup, then **Run steel design**; the members are coloured by utilization and every check is listed with its clause (Design).
8. **Report**: Report ▸ Report builder, preview, download the PDF or print (Reports).

The first time you open a model a short **tour** points at these controls; Help ▸ Tour shows it again.

Quickstart wizard

New model ▸ **Quickstart wizard…** builds a complete, loaded and analysed steel moment frame in five steps:

1. **Building**: number of stories (1 to 50), height and seismic weight of every story, bays, bay width, the number of frames that share the lateral load, column and beam W-shapes, fixed or pinned bases.
2. **System**: steel special, intermediate or ordinary moment frame (R, Cd, Ω0 from ASCE 7-22 Table 12.2-1; Ct and x from Table 12.8-2).
3. **Site and wind**: Ss, S1, site class, Ie, TL; basic wind speed, exposure and the plan dimensions.
4. **Build and run**: the wizard computes the ELF story forces (ASCE 7-22 §12.8) and the MWFRS wind story forces (Ch. 27), generates the frame, writes the dead (story weight on the beams), seismic and wind cases on the story levels, generates the LRFD combinations, saves the model in the project, analyses the seismic case and designs every member to AISC 360.
5. **Results**: base shears, the governing lateral load, the system factors, the story drift from the analysis ($\Delta = Cd \cdot \delta xe / Ie$ against the Table 12.12-1 limit) and the design summary, with **Open the model** and **Open full report (PDF)**.

The wizard's frame is a starting point: check the loads, add the gravity live load and refine the model in the editor.

Where things are

You want to…	Go to
Change units	View ▸ Units, the status bar (click to cycle), or Settings
Find any command	**Ctrl+K** (command palette)
Undo / redo	Ctrl+Z / Ctrl+Y (every model edit, including design parameters and optimize)
Save now	Ctrl+S (autosave is on)
Restore an older version	Open the model ▸ Home ▸ Versions (or Ctrl+K, "Versions")
Ask a question	Support (header) or services@ilxstudio.com

CHAPTER 3

The editor

Opening a model opens the editor. It has six parts:

Part	What it does
Ribbon (top)	Tabs Home, Model, Loads, Analyze, Results, Design, Advanced, Report, View, Seismic, Bridge and Help; the model name, the save state, the command palette, panel toggles and the theme.
Model tree (left)	Nodes, members, plates, supports, materials, sections, load cases, combinations and groups with counts; click selects in the 3D view, double-click opens the editor for the item.
3D view (centre)	The model, loads and results; selection, drawing tools and the view cube.
Properties (right)	The selection's properties, edited in place.
Tables (bottom)	Spreadsheet-style grids of the model (nodes, members, plates, supports, loads, combinations) and of the results (displacements, reactions, member forces, envelope, modal, buckling, design).
Status bar (bottom line)	Tool, selection, active case, results shown (and whether they are stale), undo label, job progress with Cancel, units.

Ribbon tabs

- **Home**: new, open, save, save as copy, export; undo, redo, delete; the select, node, member, plate and support tools; fit and zoom; templates.
- **Model**: geometry generators, grid lines and stories, split and merge, supports, sections, materials, end releases, roll angle, mirror, array, rotate, groups and the advanced model features.
- **Loads**: load cases, combinations (generate ASCE 7-22 LRFD / ASD), nodal, distributed, point, area and self-weight loads, the loads table and the wind, snow and seismic generators.
- **Analyze**: run (F5), the analysis types, the model check, cancel.
- **Results**: deformed shape, diagrams, reactions, mode shapes, plate contours, utilization colours, tables, server-shaded view, section cut.
- **Design**: steel design setup and run, design results, utilization colours, optimize, beam sizing, composite beam, concrete, shear wall, wood, masonry, AISC 341 seismic checks, story and wind drift, DAM helper.
- **Advanced**: stability, robustness, moving loads, forced vibration, staged construction, continuum and specialty analyses.
- **Report**: report builder, PDF, preview, print, results CSV, title block.
- **View**: camera views, projection, display toggles, labels, units, high contrast, text size.
- **Seismic**: modal analysis, mode animation, response spectrum, ELF.
- **Bridge**: send to ILX Connection / ILX Calcs, connection type, project navigator.
- **Help**: user manual, tour, keyboard shortcuts, About, Settings, Account.

Command palette

Ctrl+K opens a search over every command (the same entries as the ribbon and the shortcuts). Type part of a name, move with the arrow keys, run with Enter.

Saving, versions and conflicts

Edits save automatically a moment after you stop (the delay is in Settings); the indicator next to the model name shows Saved, Unsaved changes, Saving or a problem. Ctrl+S saves at once. If the same model is saved from another tab or device, a banner offers to load that version or to keep yours. Leaving the page with unsaved changes asks first. Every save is also a **version**: Home ▸ Versions lists them, saves a named version and restores an older one (the current state is kept as a version first, so a restore can itself be undone).

Jobs

Analyses, designs, reports and CSV exports run as **jobs** on the server. The status bar shows the progress and a Cancel button; the same request a second time comes back at once from the results cache. Results carry the model they belong to: after an edit they are marked **stale** until you run again.

CHAPTER 4

Modelling

Coordinates are global X, Y (horizontal) and **Z up**. The model is stored in kip and inches; every field shows and accepts the current display units (Units).

Nodes

- **Node tool (N)**: click on the work plane; the cursor snaps to the grid, nodes, midpoints and intersections. Type coordinates for an exact point.
- **Node grid generator** (Model ▸ Geometry): nx × ny nodes with spacings.
- **Tables**: type or paste coordinates into the Nodes grid (paste a block from Excel; rows past the end are added).

Members

- **Member tool (M)**: click a start node and then the end nodes of a chain; Esc ends the chain. The next member's section, material and end releases are shown in the Properties panel.
- **Section** (Model ▸ Properties): every AISC v16 shape (2,299 shapes, 14 families) with search and a family filter, or a parametric section (rectangle, round, pipe, HSS, I, channel, angle, tee).
- **Material**: steel, concrete, wood and other grades from the library, or custom values (E, G, Fy, Fu, density).
- **End releases**: presets (pinned at i, at j, truss) or single degrees of freedom in Properties. Releases are 12 flags: ux, uy, uz, rx, ry, rz at end i, then at end j.
- **Roll angle**: rotates the section about its axis; with 0° a beam's strong axis (local z) is horizontal and its local y points up; a column's local z is global X. View ▸ Local axes shows x (red, i → j), y (green) and z (blue).
- **Split** into equal parts, **merge** coincident nodes, **move, copy, mirror, array, rotate** the selection.

Plates

Plate tool (P): click 3 or 4 nodes (Enter finishes a triangle). Thickness and material in Properties. The mesh tool and the free mesher (Analyze / Advanced) make plate regions with openings.

Supports and springs

Fixed, pinned and roller presets (Model ▸ Supports) apply to the selected nodes or start the Support tool. **Custom DOF / spring** sets each of the six restraints or a spring stiffness per DOF (kip/in, kip·in/rad).

Grid lines and stories

Model ▸ **Grid lines & stories** defines plan grid lines (labels, X or Y, position) and story levels (name, elevation). Story levels drive the drift checks, the diaphragms and the ELF / wind story loads.

Groups and selection

Click selects; Shift adds, Ctrl toggles, Ctrl+Shift removes. A left drag draws a box: left to right selects what is fully inside, right to left what it crosses. **Groups & select by…** saves named selections and selects by section, material, orientation or level. Design groups (Design ▸ Setup) are separate: they carry design parameters.

Advanced model features

Rigid links (master and slaves), rigid diaphragms per story, rigid end offsets, symmetry conditions, Winkler soil springs, prestress tendons with losses, nodal masses, settlements and thermal loads (Model ▸ Advanced and the Advanced tab).

Import and export

New model ▸ Import reads `.ilxs`, DXF, IFC, STEP, IGES, ETABS `.e2k` and mesh files (gmsh `.msh`, Abaqus `.inp`, UNV). Imported members get a default material and section: assign the real ones next. Home ▸ Export downloads `.ilxs`, DXF, IFC, CSV and a PNG of the view.

CHAPTER 5

Loads and combinations

Load cases

Loads ▸ **Load cases** adds, renames and deletes cases. Each case has a type (Dead, Live, Roof Live, Snow, Rain, Wind, Seismic, Earth, Fluid, Other) and a short code (D, L, Lr, S, R, W, E, H, F, T, SW). The combination generator uses the type, so set it correctly. The **active case** (Loads tab) receives the loads you apply and is drawn in the view.

Applying loads

Load	How
Nodal load	Fx, Fy, Fz, Mx, My, Mz on the selected nodes.
Distributed load	Uniform or trapezoidal (w1, w2) on the selected members, full length or from a to b, in global X / Y / Z or member local x / y / z, optionally per projected length.
Point load on member	P at distance a from end i, global or local direction.
Area load	A pressure on plates (normal or global direction) or on members with a tributary width.
Self-weight	Member (and plate) weight from the material density × factor, in a Dead case.

The Loads table lists every load with how it reaches the engine. Gravity is −Z: a downward load is a negative Fz or wz.

Environmental load generators

Loads ▸ Environment and the Seismic tab generate ASCE 7-22 loads onto the story levels:

- **Wind** (Ch. 26 and 27, directional procedure): V, exposure, Kd, Kzt, Ke, enclosure / GCpi, G, plan dimensions, roof slope; story forces and the Fig. 27.3-8 load cases.
- **Snow** (Ch. 7): ground snow load, exposure, thermal and importance factors, slope; balanced, unbalanced, drift and sliding snow.
- **Seismic ELF** (Ch. 11 and 12): Ss, S1, site class, R, Ie, Cd, Ω0, period, seismic weight, vertical distribution, irregularities and ρ.

Combinations

Loads ▸ **Generate ASCE 7-22 LRFD** (§2.3) or **ASD** (§2.4) builds the combinations from the model's case types, with the options of §2.3.1 (companion live factor f1 0.5 or 1.0), Ev = 0.2SDS·D, ρ and reversed lateral loads. The Combinations editor adds, edits and deletes rows (one factor per case). Each combination carries its method (LRFD or ASD); the steel design uses the LRFD combinations.

Results are shown per load case (unfactored), per combination, or as the envelope of all combinations.

CHAPTER 6

Analysis

Every analysis runs as a server job with progress and Cancel. Before a run the model is checked (unsupported or floating nodes, mechanisms, zero stiffness): problems are listed and clicking one selects the item.

Run (F5)

F5 solves every combination with one factorization and builds the envelope (with member diagrams). Without combinations it solves the active load case. Large models use the GPU solver on the server when it is faster; the job says which solver ran.

Analysis types

Analysis	Where	What you enter	What you get
Linear static	Analyze ▸ Linear static	all combinations, one combination, or one load case; diagrams; solver	displacements, reactions, member end forces, diagram stations
P-Δ (second order)	Analyze ▸ P-Δ	combination, iterations, tolerance	second-order results of that combination
Modal	Analyze ▸ Modal, Seismic ▸ Modal	number of modes; mass from density and nodal masses	periods, frequencies, participation (90 % check, ASCE 7 §12.9.1), mode shapes
Buckling	Analyze ▸ Buckling	reference loads, modes	critical load factors λ and buckled shapes
Response spectrum	Seismic ▸ Response spectrum	ASCE 7-22 spectrum (Ss, S1, site class, TL), direction, modes, damping, CQC / SRSS	modal and combined base shear, story demands
Time history	Analyze / Advanced	ground motion (upload), scale, direction, modal or Newmark, damping	response histories
Pushover	Analyze / Advanced	load pattern, control node, target, hinges (or ASCE 41 hinges)	capacity curve, hinge sequence, performance point
Nonlinear buckling, arc length	Advanced ▸ Stability	imperfection, steps	load-displacement path
Progressive collapse	Advanced ▸ Robustness	members to remove, combination	alternate-path DCRs
Fatigue	Advanced ▸ Robustness	stress history, detail category	rainflow and damage
Influence lines	Advanced ▸ Moving	path, response	influence line
Floor vibration	Advanced ▸ Service	fn, W, β, occupancy (AISC DG11)	peak acceleration vs limit

Analysis	Where	What you enter	What you get
Harmonic, transient, PSD	Advanced ▸ Forced vibration	load, frequencies or PSD table, damping	response functions
Staged construction	Advanced ▸ Construction	stages	results per stage
Contact, material nonlinear	Advanced ▸ Specialty	gaps or material laws	nonlinear results
Continuum, thermal, fracture	Advanced ▸ Continuum	mesh, material, loads	contour results
Section cut	Results ▸ Section cut	plane, combination	resultant forces through the cut
Story drift, deflection	Design ▸ Story drift / Wind drift, Analyze ▸ Drift	levels, Cd, Ie, limits	drift checks (Design)

Second-order effects

A first-order analysis does not include P-Δ. For design, choose the second-order method in Design ▸ Setup (P-Δ, the direct analysis method or B1-B2): the design job then analyses every design combination that way. Analyze ▸ P-Δ is for looking at one combination's second-order response.

Stale results

Results belong to the model they were computed for. After any edit the status bar shows **(stale)**; press F5 again. The design results are stale in the same way.

CHAPTER 7

Results

Choose what to show with **Results for** on the Results tab: a combination, the envelope of all combinations, a P-Δ run, or one load case (solved on demand, unfactored).

In the 3D view

- **Deformed shape** with a scale slider (automatic scale × factor). Members are drawn as curves from the end displacements and rotations.
- **Diagrams** N, Vy, Vz, T, My, Mz along every member, with a scale slider; for the envelope the maximum and minimum are drawn.
- **Reactions** as arrows with values.
- **Mode shapes** (modal and buckling) with an animation player and a mode picker.
- **Plate contours** of the displacements.
- **Utilization colours** from the design: green up to the first band (0.90), amber up to 1.00, red above, grey not designed. The bands are in Design ▸ Setup and Settings.
- **Server shaded view**: shaded members with the chosen diagram, rendered by the engine.

Hovering a member shows its id, section and, with design results, its ratio.

Tables

The bottom panel has result grids for displacements, reactions (with totals and the residual), member end forces, the envelope (max / min with the governing combination), modal participation and buckling factors, plus the **Design** table. Copy rows with Ctrl+C or download CSV.

Signs

Member end forces are in member local axes: N (tension +), Vy, Vz, T, My, Mz. On a horizontal beam a sagging moment is a negative Mz (a positive Mz puts the +y, upper, fibre in tension). Reactions are the forces the supports exert on the structure; a gravity load gives a positive Fz. See Units and sign conventions.

CHAPTER 8

Design

The Design tab checks the model's own members with the forces of the combinations you choose, analysed with the second-order method you choose. Nothing is checked against a fixed section or a demo load.

Steel design (AISC 360, LRFD)

Design setup

Design ▸ **Design setup** has four pages. Everything is saved with the model and undoable.

- **Method**: LRFD (ϕRn against the §2.3 combinations) or ASD (Rn/Ω, AISC 360-16 §B3.2, against the §2.4 combinations), the **second-order method** and its options, the model-wide K and the utilization colour bands:

Method	What the design job does
First order	Linear analysis of every design combination; the B1 member amplifier (App. 8 §8.2.1) on members in compression when on. Sway effects are not included.
P-Δ	Geometric-stiffness P-Δ analysis of every design combination (ASD combinations at 1.6× and scaled back).
Direct analysis method	AISC 360-22 Ch. C: 0.8 EI and EA, τb (per member or with additional notional load), notional loads Ni = 0.002 Yi (automatic: gravity-only combinations), P-Δ; K = 1 (Lc = L) for every member.
B1-B2	Approximate second-order analysis (App. 8): B2 per story with RM (0.85 moment frames, 1.0 braced).

- **Combinations**: the design combinations (default: every LRFD combination). Only these are analysed and enveloped for design.
- **Member parameters**: per member the **role** (beam, column, brace, or automatic from the slope), **K**, **Lb** (unbraced length for lateral-torsional buckling; blank = member length), **Cb** (AISC 360 F1; blank = 1.0, conservative), **Lcx** and **Lcy** (effective lengths for flexural buckling about the strong and weak axes; blank = K·L). Set values for the whole selection at once, or per row.
- **Design groups**: named sets of members with shared role and lengths. A member's own values override its group; the group overrides the model-wide values.

Run and results

Design ▸ **Run steel design** runs the job, colours the members by utilization and opens the **Design** table in the bottom panel:

- one row per member: section, role, length, ratio, status, governing check, clause and combination;
- sort by any column, filter by text, status (over capacity, not certified, OK) and role, download CSV;
- a click selects the member and shows its **detail**: the parameters used (K, Lb, Cb, Lc) and every check with demand, capacity, ratio, clause, combination and the engine's step-by-step detail.

The checks: D2 tension, E3 / E4 compression (E7 slender elements), F2 / F3 / F5 strong-axis flexure with Lb and Cb, F6 weak-axis flexure, G2 shear and the H1-1a / H1-1b interaction. Hover a clause for what it checks.

Not certified

When the engine cannot compute a capacity conservatively for a section (for example a slender element whose dimensions are unknown, or Pr ≥ Pe1 so no B1 can be formed), the check is **not certified**. A not-certified member is shown with its own status and is **never counted as passing**: check it by hand or use a section the engine can check.

Optimize

Design ▸ **Optimize members** finds the lightest catalogue W-shape whose every check is certified and at most the target ratio (default 0.95), for the selected members, a design group or all members, with the same combinations and second-order method. Review the table (current → proposed, ratio), untick any change, then **Accept**: all changes are one undo step. The member forces come from the current analysis: run the analysis and design again after accepting, because the stiffness changed.

Size a beam

Design ▸ **Size a beam** lists every W-shape that passes F2, G2 and an L/360 live-load deflection check for Mu, Vu, span, Lb, Cb, Fy and optional depth and weight limits; **Demands from member** takes Mu, Vu and L from the analysis envelope of the selected member. **Assign** puts the section on the member (undoable).

Composite beam (AISC 360 Ch. I)

Span, W-shape, Fy, beam spacing, deck rib height and orientation, concrete above the deck, f'c, unit weight, stud diameter, studs per side (blank = full composite), factored load and service live load. Results: ϕMn vs Mu, stud count and composite ratio, effective width, lower-bound inertia, live-load deflection and floor frequency, with the engine's worked summary.

Concrete (ACI 318-19)

- **Beam**: b, d, As, stirrups Av at s, f'c, fy, λ; Mu and Vu. Checks §22.3 flexure (ϕ from εt, §21.2), §9.6.1 minimum steel, maximum steel (εt ≥ 0.005), §22.5 shear.
- **Column**: Ag, Ast, f'c, fy, Pu; for slenderness Ig, r, k, Lu, βdns, M1, M2 and transverse loading. Checks §22.4 axial strength and the §6.6.4 moment magnification.
- **Shear wall**: lw, thickness, hw, ρl, ρt, f'c, fy, λ, ϕ; Pu, Mu, Vu. Checks §11.5.4 in-plane shear, §11.6 minimum reinforcement, §11.5.3 axial force and flexure.

Wood (NDS)

- **Beam**: b, d, Fb, Emin, Le, adjustment factors CD, CM, Ct, CF (blank = size factor from d), Ci, Cr, Cfu; M. Checks §3.3 bending with CL (fully braced when Le = 0).
- **Column**: b, d, Fc, E, Emin, Le, KcE, c, factors; P. Checks §3.7 compression with CP.

Enter ASD-level demands for wood and masonry.

Masonry (TMS 402-16)

Wall strip: thickness and width (computes Ag, Sg, r, In), effective height, f'm, Em, eccentricity, P, M. Checks §8.2 axial and flexural stresses, the unity check Eq. 8-11 and the ¼Pe buckling limit Eq. 8-16.

Seismic design (AISC 341-22)

Design ▸ **Seismic (AISC 341)**:

- **Special moment frame** (§E3): beam and column W-shapes, Fy, Fu, Ry, clear span, Puc, gravity shear, column shear, connection type (RBS with a, b, c; WUF-W; others with sh), joint layout, doubler plate. Checks strong column / weak beam (Eq. E3-1), panel-zone shear and thickness, continuity plates.
- **Special concentrically braced frame** (§F2): brace shape, material (Ry from Table A3.2), Lc, element and b/t, Pu, Tu, the tension-brace share of the lateral force. Checks slenderness, width-to-thickness (λhd), strength and the force distribution, and gives the capacity-limited brace forces for the connections.
- **Eccentrically braced frame link** (§F3): link W-shape, Fy, e, Vu, Pr, Mp, link rotation. Checks the link class, shear strength, length limit and rotation.

System R, Cd, Ω0 gives the ASCE 7-22 Table 12.2-1 factors and the Table 12.12-1 drift limit for the system, risk category, structure type, ρ and SDC.

Story drift and wind drift

Design ▸ **Story drift** (ASCE 7-22 §12.12, §12.8.7, Table 12.3-2): choose the solved case or combination with the seismic loads, the direction, Cd, Ie, ρ and the risk category, then **Fill from the analysis**: the stories come from Model ▸ Grid lines & stories (or the node elevations), δx and δxe from the analysis, V and P from the loads above each story (editable). The check gives Δ = Cd·δxe/Ie against Δa, θ against θmax, and the vertical irregularities.

Design ▸ **Wind drift**: a service-level wind result, the total (H/400) and story (h/600) limits.

Direct analysis method helper

Design ▸ **DAM helper** computes the notional loads Ni for a list of story gravity loads, τb for Pr and Py, and the Δ2nd / Δ1st ratio. The direct analysis method itself is a second-order method in Design setup.

CHAPTER 9

Reports

Report ▸ **Report builder** makes the calculation report of the current model as a server job.

Content

Choose the parts:

Part	Contents
Model summary	counts, materials, sections, solver health
Inputs	nodes, members, supports as the engine consumed them
Load cases and combinations	every case and the combination factors
Support reactions	reactions per support
Member forces	end forces and the governing combination per component
Force diagrams	moment and shear diagrams of the governing members
Design checks	AISC 360 checks with clauses, limit-state detail and substituted values
Conventions and glossary	axes, units, signs and terms

The title block, the status badge and the disclaimer with the engine version are always printed. **Use the Design tab settings** makes the report's checks the same as the Design tab's (design combinations, member parameters, second-order method). The choices are saved with the model.

Project information and title block

- **Project**: name, job number, client, site address, city, engineer and issue date, shared by every model of the project.
- **Title block**: firm, address, phone, licence, engineer of record and licence, prepared and checked by, and the **status** printed in the badge (DRAFT until you choose an issued status such as FOR REVIEW or ISSUED FOR CONSTRUCTION). Set your defaults once (My defaults); a project can override any field (This project), and Reset makes it inherit again.

Output

- **Preview** renders the HTML report in the panel.
- **Download PDF** saves the PDF report.
- **Print** (Ctrl+P in the editor) opens the browser's print dialog for the report.
- **Results CSV** downloads the reactions, member forces, combinations and per-combination design demands.

Results are computed for the review of a licensed professional engineer, who remains responsible for the design.

CHAPTER 10

Bridge to ILX Connection and ILX Calcs

The Bridge tab hands a member and its forces to the other ILX web apps through your ILX account.

Send to ILX Connection / ILX Calcs

1. Select a member (or pick it in the dialog).
2. Bridge ▸ **Send to Connection** (ilxconnection.com) or **Send to Calcs** (ilxcalcs.com).
3. Choose the **end** (i or j), the **combinations** (all, up to 60, or one) and, for Connection, the **connection type**: single plate, double angle, shear end plate, shear tee, seated, flange-plated moment, extended end-plate moment, directly welded moment, or the column base plate.
4. **Build the link**: the model is saved first; the server solves every combination (or uses the cached results) and builds the payload. The dialog lists the end forces per combination (member local axes, N tension +).
5. **Open in ILX Connection / ILX Calcs**: a new, unsaved calculation opens with the section, grade and the forces (V and P for shear connections, M as well for moment connections) for you to review before saving.

The data travels in the link's address fragment, which is never sent to a server or written to a log.

Connection type per member end

Bridge ▸ **Connection type**: for the selected members and end (i, j or both) choose **Shear connection** (the end moments My and Mz are released in the analysis) or **Moment connection** (restored). The choice is stored on the member and used by the hand-off; Undo reverts it.

Project navigator

Bridge ▸ **Project navigator** lists the models of the project (open any of them) and shows how many members of this model are designed. The model tree (left) is the navigator of the open model.

CHAPTER 11

Units and sign conventions

Units

The model is stored and solved in **kip, inch, second, ksi and radian**. What you see and type follows the unit system chosen in View ▸ Units (or the status bar, or Settings):

Quantity	kip, in	kip, ft	kN, m (SI)
Coordinates, lengths	in	ft	m
Section dimensions, thickness	in	in	mm
Force	kip	kip	kN
Moment	kip·in	kip·ft	kN·m
Line load	kip/in	kip/ft	kN/m
Pressure	psf	psf	kPa
Stress, E, Fy	ksi	ksi	MPa
Displacement	in	in	mm
Rotation	rad	rad	rad
Roll angle	°	°	°

Number fields accept arithmetic (`12*3.5`, `(20+4)/2`) and, in US units, feet and inches (`12'6"`, `12' 6-1/2"`). Each model remembers its unit system; new models use your default (Settings).

Axes

- **Global**: right-handed X, Y, Z with **Z up**; gravity acts in −Z. Nodal loads, displacements and reactions are global.
- **Member local**: x runs from end i to end j; **z is the strong axis** (Iz = AISC Ix) and y the weak axis. With roll 0:

Member	local z	local y
Not vertical	x × Z, horizontal	z × x, points upward
Vertical	global X (made orthogonal to x)	z × x

  So a beam along +X has y = +Z (up) and z = −Y; a column drawn upward has z = +X and y = −Y. The roll angle turns y and z about x by the right-hand rule. View ▸ Local axes draws them.
- **Plates**: the normal (the "top" face) follows the right-hand rule on the node order; counter-clockwise seen from above points up. Reporting axis 1 is global X projected onto the plate (global Y when the plate is nearly normal to X).

Member end forces

`(N, Vy, Vz, T, My, Mz)` at each end in local axes. **N is tension positive.** Vy, Vz, My, Mz at an end are the force and moment the joint applies to the member, positive along / about +y and +z; so a simply supported beam under gravity has Vy = +wL/2 at both ends.

Diagrams, sagging and hogging

The diagrams give the internal forces at each station (0 at end i to L at end j); dMz/dx = −Vy and dMy/dx = +Vz.

- A **positive Mz puts the +y fibre in tension**; a positive My puts the −z fibre in tension.
- On a horizontal beam (local y up), a **sagging** moment (bottom in tension) is **negative Mz**: a simple span under w has Mz(L/2) = −wL²/8. A **hogging** moment is positive: a fixed-end beam has Mz = +wL²/12 at the ends.
- A downward deflection of such a beam is a negative local uy.

Reactions and loads

- Reactions `(Fx, Fy, Fz, Mx, My, Mz)` are global: the force the support applies to the structure. A gravity load gives Fz > 0.
- Loads act along the chosen axis: a gravity load is negative Z (global) or negative y (local, on a horizontal beam).

Plate and solid results

- Membrane forces Nxx, Nyy, Nxy (kip/in): tension positive.
- Plate moments Mxx, Myy, Mxy (kip·in/in): **positive = tension on the bottom face**, so an upward-facing slab shows sagging as positive.
- Surface stresses σtop = N/t − 6M/t², σbot = N/t + 6M/t²; von Mises and principal stresses (σ1 ≥ σ2).
- Solid stresses are global, tension positive.

Design ratios

Ratio = required strength / design strength (ϕRn). Up to 1.00 passes; above 1.00 does not. **Not certified** means the engine could not check that limit state conservatively: it is not a pass.

Codes

AISC 360-16 (member checks labelled with their clauses, LRFD and ASD; the direct analysis method of Ch. C and the B1-B2 method of App. 8 as in AISC 360-22), AISC 341-22 and AISC 358-22, ACI 318-19, NDS 2018 (ASD), TMS 402-16, ASCE 7-22 (loads, combinations, drift) and ASCE 41-23 (pushover hinges). Help ▸ About lists the same editions.

CHAPTER 12

Settings and account

Settings

Help ▸ **Settings** (or the menu in the header) has two scopes:

- **My defaults**: stored with your account, the same on every device, used by every project that does not override them.
- **This project**: overrides the units and the design defaults for one project. A field you set shows "project"; **Reset** makes it inherit your default again.

Page	Settings
General	default units, autosave delay, text size, theme (System / Light / Dark)
Design defaults	design method, second-order method, default K, B1 amplifier, utilization bands, optimize target
Display	default display toggles (extruded sections, labels, local axes, loads, grid, axes) and the snap grid
Privacy	what usage information is recorded

A model's own choices (View ▸ Units, Design ▸ Setup) take precedence over the defaults. The title block and report defaults are in Report ▸ Title block, with the same My defaults → project inheritance.

Account

Help ▸ **Account** (or your name in the header) shows who is signed in and your licence, with links to **My ILX account** (my.ilxstudio.com: profile, subscription, billing) and **Manage subscription**, and **Sign out**. ILX Structures has one plan with every feature. When a licence has ended the app is read-only: you can open, view and download your models but not change them or start jobs.

Support

The Support button in the header opens a ticket (question, problem, suggestion, billing); answers arrive in the app and by email. You can also write to services@ilxstudio.com.

CHAPTER 13

Keyboard and mouse

Keyboard

Keys	Action
Ctrl+K	Command palette (every command)
Alt+N	New model (Ctrl+N is reserved by browsers)
Ctrl+O	Open model
Ctrl+S	Save now (autosave is on)
Ctrl+Shift+S	Save as copy
Ctrl+Z	Undo
Ctrl+Y, Ctrl+Shift+Z	Redo
F5	Run analysis (every combination with the envelope)
Ctrl+P	Print the report
F1	User manual
Del, Backspace	Delete the selection (nodes take their members with them)
Esc	End the member chain, leave the tool, clear the selection, close a dialog
Ctrl+A	Select all
N, M, P, S	Node, member, plate, select tool
F	Fit the view
Z	Zoom to the selection
+ / −	Zoom
L	Load on the selection: nodal load (nodes), distributed load (members), area load (plates)
Ctrl+D	Duplicate the selection (Copy with an offset)
Ctrl+R	Run the steel design (AISC 360)
G, R	Move gizmo, rotate gizmo
Tab, Shift+Tab	In the 3D view: select the next / previous node, member or plate
Numpad 7, 1, 3, 0	Top, front, right, iso view; with Ctrl: bottom, back, left. Numpad 5: orthographic; 4 / 6 orbit; 8 / 2 tilt; . zoom to the selection
Enter	Finish a triangle plate; confirm a dialog (runs its main button); finish a pick
Backspace (pick mode)	Remove the last picked point

In the tables: arrows move, type or F2 / Enter edits, Ctrl+C copies the selected rows, Ctrl+V pastes a block from a spreadsheet.

Mouse

Gesture	Action
Click	Select (Shift adds, Ctrl toggles, Ctrl+Shift removes)
Left drag	Box select: left to right = inside the box, right to left = crossing
Right drag	Orbit (View ▸ Left-drag orbits for trackpads)
Middle drag, Shift + right drag	Pan
Wheel, pinch	Zoom to the cursor
Double-click empty space	Fit the view
Shift + wheel	Turntable: turn the model about the vertical axis, 10° per notch (View ▸ Viewport settings)
View cube	Click a face for that view
Nav ring	Arrows around the cube orbit 45° and tilt 30°; ⟳ spins the model like a turntable
Pick mode	A form's **Pick** button: click nodes (vertex), points on members (edge) or plates (face) in the view; Enter finishes, Esc cancels

CHAPTER 14

Troubleshooting

Message or symptom	Cause and fix
"The model has N errors; fix them to run"	The model check found unsupported or floating nodes, a mechanism or members without stiffness. Click a problem to select the item; add supports or releases accordingly.
Unstable / unconverged solve in the report	The structure is a mechanism or nearly so (residual too large). Check supports, end releases (a member released at both ends in rotation and torsion is a mechanism) and plate connectivity.
Results marked "(stale)"	The model changed after the run. Press F5 (and run the design again).
A member is "Not certified"	A capacity cannot be computed conservatively for that section (missing element dimensions, slender element, Pr ≥ Pe1). Check it by hand, change the section, or use a second-order method.
"No LRFD combination is selected for design"	Design ▸ Setup ▸ Combinations: tick the combinations, or generate ASCE 7-22 LRFD combinations in Loads.
ASD steel design	The steel checks are LRFD only for now; use LRFD combinations. Wood and masonry checks take ASD-level demands.
Drift check shows zeros	Choose a result with lateral loads (the seismic case or combination) and the right direction, then Fill from the analysis. Define story levels in Model ▸ Grid lines & stories.
Hand-off: "model could not be saved"	The hand-off reads the saved model. Resolve the save problem (conflict banner or connection) and try again.
"Another job is running"	Each person runs up to two jobs at a time; wait for it or cancel it in the status bar.
Read-only banner	Your licence has ended or the licensing service is unreachable. Projects stay viewable; Subscribe or Check again.
"This model changed somewhere else"	It was saved from another tab or device. Load the saved version or keep yours (overwrite).

Still stuck? Open a Support ticket from the header, or write to services@ilxstudio.com with the project and model name.



ILX **Structures**

3D structural analysis and design in your browser

https://ilxstructures.com · services@ilxstudio.com

© 2026 ILX Studio, LLC