

EDFS Matrix

Manual & Technical Reference

User guide and technical reference for the analysis & design of 3D frame structures

Application version	1.0.0
Document date	19 October 2026
Codes	SNI 1729:2020 · SNI 2847:2019 · SNI 1727:2020 · SNI 1726:2019
Publisher	Edifisia

Contents

Introduction	4
About this document	4
How to read the cross-references	4
Codes and standards	4
User responsibility	4
Part A — User Manual	5
M1. Getting to know EDFS Matrix	6
M1.1 What it does	6
M1.2 Account and access	6
M1.3 Main window	7
M1.4 General workflow	7
M1.5 Units	8
M2. File menu	9
M2.1 Project: New, Open, Save, Save As	9
M2.2 Exchange: Import and Export	9
M2.3 Report	9
M2.4 Account: Sign In / Sign Out	10
M3. Model menu	11
M3.1 History and Select	11
M3.2 Draw	11
M3.3 Geometry: Node/Frame Tables	11
M3.4 Modify	11
M4. View menu	13
M5. Definition menu	14
M5.1 Geometry: Grid and Groups	14
M5.2 Properties: Materials, Frame Sections, Shell Sections	14
M5.3 Joint: Constraints	14
M5.4 Loads: Load Cases, Load Combos, Combo Groups, Seismic	14
M5.5 Mass: Mass Source	15
M6. Assign menu	16
M6.1 Joint	16
M6.2 Frame	16
M6.3 Area	16
M7. Run menu	17
M7.1 Check Model	17
M7.2 Run... and Run	17
M8. Output menu	18
M8.1 Shape	18
M8.2 Member Forces	18
M8.3 Tables: Result Tables and Material Take Off	18
M9. Design menu	19
M9.1 Design Parameters	19
M9.2 Check Design	19
M9.3 Display and Detail	19
M10. Help menu, navigation and panels	20

M10.1 Help menu	20
M10.2 Shortcuts and mouse	20
M10.3 Properties panel and tables	20
M11. Example: 3D concrete frame	21
Part B — Technical Reference	22
R1. Model and elements	23
R1.1 Units	23
R1.2 Axis systems	23
R1.3 3D frame element	23
R1.4 Section properties	23
R1.5 Property modifiers	24
R1.6 Analysis model (discretisation)	24
R1.7 End releases	24
R1.8 Joint constraints	25
R1.9 Supports and support springs	25
R1.10 End offsets and insertion points	25
R2. Loads and combinations	27
R2.1 Frame loads	27
R2.2 Loads per projected length	27
R2.3 Self weight	27
R2.4 Load case types	27
R2.5 Notional loads (SNI 1729 § C2.2b)	27
R2.6 Mass source	28
R2.7 Load combinations	28
R2.8 Load-distributing areas	30
R2.9 Combination groups and envelopes	30
R3. Earthquake loads to SNI 1726:2019	31
R3.1 Parameters and design spectrum	31
R3.2 Lateral load of a Quake load case	31
R3.3 Seismic weight and force distribution	31
R3.4 Response spectrum	31
R3.5 Not yet available	31
R4. Linear analysis	32
R4.1 Linear static analysis	32
R4.2 Member forces and sign convention	32
R4.3 Modal analysis	32
R4.4 Diaphragm centre of mass and centre of rigidity	33
R5. Non-linear analysis	34
R5.1 P-Delta (second order)	34
R5.2 Direct analysis method — SNI 1729:2020 Chapter C	34
R5.3 Second order for concrete — SNI 2847:2019 § 6.7	35
R5.4 Tension-only / compression-only	35
R5.5 Run options	35
R6. Steel design to SNI 1729:2020	36
R6.1 Workflow	36
R6.2 Parameters and unbraced lengths	36
R6.3 Strength per section type	36
R6.4 Assumptions per section type	37
R6.5 Not checked	37
R6.6 Second-order effects and design	37

R6.7 ASD	38
R7. Concrete design to SNI 2847:2019	39
R7.1 Workflow and parameters	39
R7.2 Beams	39
R7.3 Columns	39
R7.4 Provided bars and drawings	40
R7.5 Capacity design (IMF / SMF)	40
R7.6 Second-order effects	40
R8. Summary of limitations, validation and verification	41
R8.1 Main assumptions and limitations	41
R8.2 Validation	41
R8.3 Verification against commercial analysis programs	41

Introduction

About this document

This document is the user guide and technical reference for **EDFS Matrix 1.0**, Edifisia's web-based application for the analysis and design of 3D frame structures. It has two parts that refer to each other:

Part	Content	For
A. User Manual (chapters M1–M11)	The main window, every menu and command starting from the File menu, navigation, and one complete example	All users; read in order when using the application for the first time
B. Technical Reference (chapters R1–R8)	Theory, solver methods, design formulas, assumptions and limitations	Engineers who need to know <i>how</i> the numbers are computed and <i>when</i> the results may be used

The manual and the technical reference are combined in one document because almost every command in the manual has theory behind it that should be known before its results are used; in one document the cross-references can be clicked directly.

How to read the cross-references

- References to the technical reference are written → **R1.7** (**chapter R1**, section 7). In the PDF they can be clicked.
- References to the manual are written → **M6.3**.
- Menu and command names are written in bold exactly as on screen, e.g. **Assign → Frame → Releases**.

Codes and standards

Code	Used for
SNI 1729:2020 (equivalent to AISC 360-16)	Steel design LRFD/ASD, direct analysis method (Chapter C), effective length method (Appendix 7)
SNI 2847:2019 (equivalent to ACI 318-14; some clauses of ACI 318-19)	Concrete design, cracked stiffness (§ 6.6.3.1.1), second-order analysis (§ 6.7)
SNI 1727:2020	Strength (cl. 2.3) and service (cl. 2.4) load combinations
SNI 1726:2019	Seismic terms in combinations, design spectrum, equivalent lateral force (§ 7.8), modal response spectrum (§ 7.9.1)
PIP STC01015 (April 2017) Tables 4 & 5	Load combinations for industrial structures (buildings & open frames)
JIS G 3192 / G 3350 / G 3466 / G 3444	Steel section database

Indonesian frame-system names are used with their equivalents: **SRPMB** = ordinary moment frame (OMF), **SRPMM** = intermediate moment frame (IMF), **SRPMK** = special moment frame (SMF); **HBK** = beam-column joint.

User responsibility

EDFS Matrix is a calculation aid. The correctness of the input, the suitability of the model for the real structure, the checking of results and design decisions remain the responsibility of the qualified professional who uses and approves the results. The program's assumptions and limitations are summarised in **chapter R8** and in the application under **Help → Legal → Batasan Teknis** (technical limitations, in Indonesian). Numerical evidence (comparisons with closed-form solutions and with commercial analysis programs) is given in the separate report *Analysis Verification EDFS Matrix* (→ **R8.3**).

Part A — User Manual

M1. Getting to know EDFS Matrix

M1.1 What it does

EDFS Matrix models and analyses **3D frame structures** (beams, columns, braces, trusses) directly in the browser, then designs the members to SNI:

- **Analysis:** linear static for all load cases and combinations, modal analysis, P-Delta (second order), the direct analysis method of SNI 1729 Chapter C, tension-only/compression-only members, earthquake loads to SNI 1726 (equivalent lateral force and response spectrum).
- **Modelling:** 3D frames with releases, end offsets and insertion points, property modifiers, joint constraints (diaphragms), supports and support springs, slab/wall areas that distribute loads to the beams.
- **Design:** steel to SNI 1729:2020 (LRFD/ASD, D/C ratio check) and concrete to SNI 2847:2019 (required reinforcement, bar and stirrup output, OMF/IMF/SMF).
- **Output:** diagrams and result tables (.xlsx export), Material Take Off, PDF reports (summary, sketch drawings, full report), file exchange as .json, .s2k (text tables) and .std (input file).

Features deliberately left out (meshed plate elements, material non-linearity, and others) are summarised in [R8.1](#).

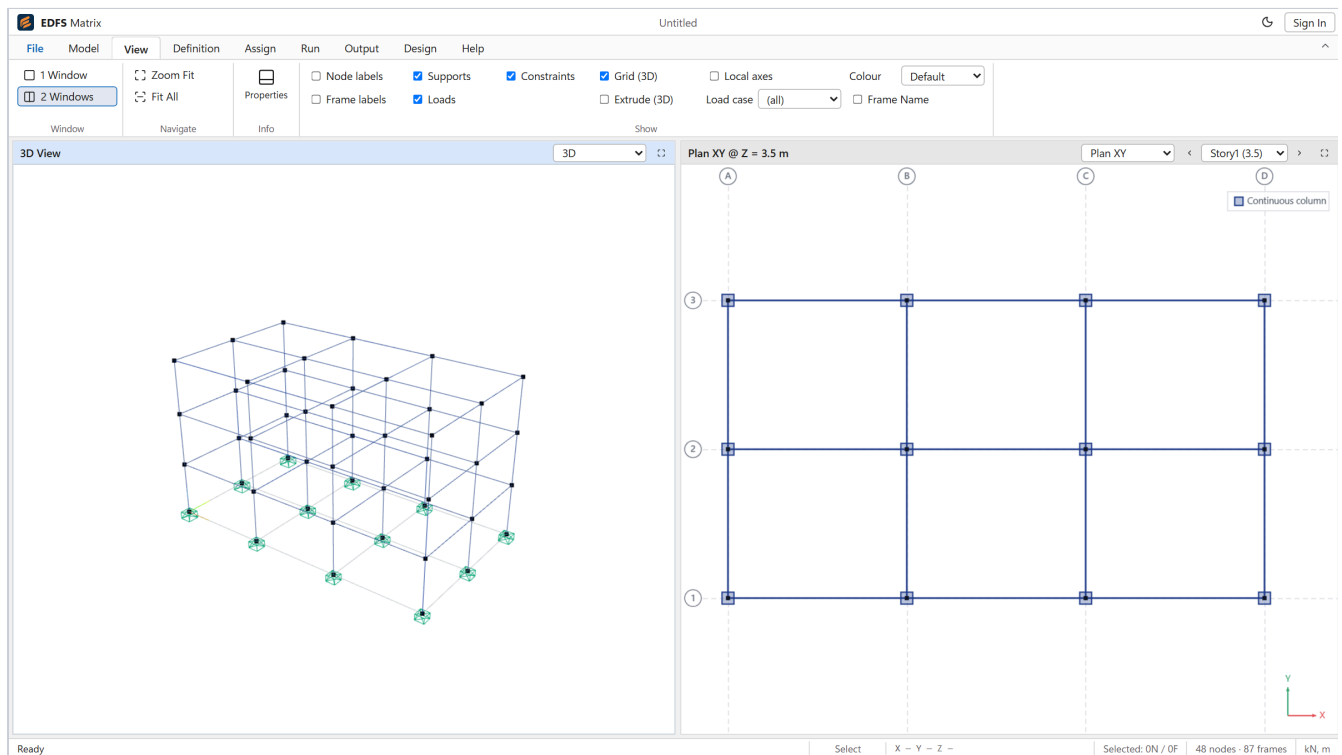
M1.2 Account and access

The application runs in a browser (latest Chrome or Edge) and can be installed as an app (PWA) through the *Install* icon in the address bar. Modelling runs in the browser; analysis runs on the Edifisia server.

Capability	Guest (not signed in)	Account without Premium	Edifisia Premium (incl. trial)
Modelling (draw, edit, assign, define), Import/Export of local files	✓	✓	✓
Run analysis and the server stage of Check Model	✗	✗	✓
Save / Save As to the cloud	✗	✗	✓
Open (read-only), export and delete own projects	—	✓	✓

A new account gets a **7-day Premium trial**. One Edifisia Premium unlocks all Edifisia applications. The model being edited is always kept as a *draft* in the browser, so reloading the page does not lose work; analysis results are not stored locally.

M1.3 Main window



The EDFS Matrix main window: ribbon (top), two views (3D and Plan), status bar (bottom).

Part	Function
Title bar	Project name, light/dark theme button, Sign In / account status
Ribbon	Nine menu tabs: File , Model , View , Definition , Assign , Run , Output , Design , Help . Double-click a tab name to hide/show the ribbon
Views	One or two windows; the view type is chosen in the window header: 3D , Plan XY , Elevation XZ , Elevation YZ . In 2D views the level/grid line is chosen with the dropdown and the ◀ ▶ buttons
Bottom panel	Properties tab (selected object, editable), Nodes and Frames (geometry tables), and the analysis and design result tables
Status bar	Last message, active command, cursor coordinates, number of selected objects, number of nodes/frames, and the model units (click to change the units)

M1.4 General workflow

1. **File** → **New**: units, quick grid, 3D frame template or grid only (**M2.1**).
2. **Definition**: grid, materials, sections, load cases, combinations, mass source, seismic parameters (**M5**).
3. **Model**: draw or edit frames and areas (**M3**).
4. **Assign**: supports, sections, releases, loads, diaphragm constraints (**M6**).
5. **Run** → **Check Model**, then **Run** (**M7**). Analysis needs Premium.
6. **Output**: deformed shape, member force diagrams, result tables (**M8**).
7. **Design**: design parameters then **Check Design**; review ratios, reinforcement and **Detail** (**M9**).
8. **File** → **Report** for a PDF report; **File** → **Save** stores the project in the cloud (**M2**).

Editing the model after a Run clears the analysis results (results always match the model on screen). Changing design parameters, combination groups or display options does not clear the results.

M1.5 Units

Units are chosen in **File** → **New** (length m or mm; force N or kN) and can be changed at any time through **Help** → **Settings** → **Units** or by clicking the units label in the status bar: every model value is converted (can be undone) and analysis results are cleared. Material strengths (F_y , F_u , f_c' , f_y) are always entered in MPa. Solver units → [R1.1](#).

M2. File menu

File	Model	View	Definition	Assign	Run	Output	Design	Help
								
New	Open	Save	Save As	Import...	Export...	Report...	Sign In	
Project				Exchange		Report	Account	

File tab.

M2.1 Project: New, Open, Save, Save As

- **New** (Ctrl+N) — *New Model* dialog: **Units** (length, force), **Quick grid** (number of lines × spacing in X, Y and Z levels) and **Template**: *3D frame on the grid* (columns S2, beams S1, fixed supports at the base) or *Grid only* (draw the members yourself). An unsaved model is replaced — the dialog warns about it.
- **Open** — list of the account's cloud projects with their versions; projects can be opened or deleted here.
- **Save** (Ctrl+S) — stores a new version of the project in the cloud (Premium). A Run of a saved project is recorded with its version. A project keeps up to its last 20 versions.
- **Save As** — saves as a new project under another name.

M2.2 Exchange: Import and Export

- **Import...** — opens a file from the computer; the format is recognised from its extension:
 - **.json** — the EDFS Matrix model format (complete, including design parameters and report information);
 - **.s2k / \$.2k** — analysis-model text tables (an exchange format common to commercial analysis programs);
 - **.std** — input file of frame analysis programs. Imported coordinates are rounded to 0.1 mm automatically. Steel sections are matched to the JIS catalogue when their dimensions agree.
- **Export...** — choose one of the same formats. After export the dialog lists what could not be carried over to that format (e.g. automatically generated earthquake cases, design preferences, springs in **.std**), so that the model can be completed in the receiving program. An exported **.s2k** file already contains the settings that make the receiving program repeat the EDFS Matrix assumptions (shear deformation off, 6-segment division, self weight per load case).

The **.s2k** export/import has been tested in both directions (→ [R8.3](#)).

M2.3 Report

Report... creates a PDF report. The first dialog asks for project information (project name, location, structure, engineer, revision — required; client, company, checker, date — optional), stored in the model. Then choose the report type and language (Indonesian/English):

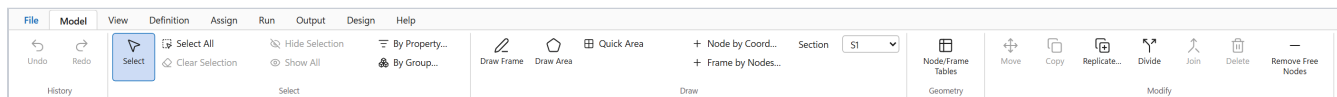
Type	Content
Summary	Project data, model, 3D picture, natural periods, total reactions, maximum displacements, design summary (1–2 pages)
Sketch drawings	3D view, plan of every level, elevation of every grid line with sections, dimensions and a title block (A3/A4)
Full report	Cover, contents, introduction → model → materials & sections → loading → analysis results → design → conclusion; Appendix A detailed calculation of every steel and concrete frame, Appendix B sketch drawings

The report is shown as a preview and printed with the browser's **Print / Save as PDF**. The report logo can be replaced (**Logo → Change...**) and is stored per browser. Reports with results need a Run first.

M2.4 Account: Sign In / Sign Out

Sign in with an Edifisia account. The Premium status is shown in the title bar. Run and Save ask you to sign in; when Premium is not active, the *Premium Access* dialog shows its expiry date.

M3. Model menu



Model tab.

M3.1 History and Select

- **Undo** (Ctrl+Z) / **Redo** (Ctrl+Y) — every model change can be undone.
- **Select** (Space) — selection mode. Click an object; **Shift/Ctrl + click** adds to the selection. Drag a box left→right (*window*: objects must be fully inside) or right→left (*crossing*: touching is enough). Double-clicking a node/frame opens the **Properties** panel.
- **Select All** (Ctrl+A), **Clear Selection** (Esc).
- **Hide Selection / Show All** — hides the selected objects from all views (display only; they are still analysed and still listed in the tables).
- **By Property...** — selects frames by section, orientation (column/beam/inclined) and Z level. Between lists the rule is AND, within a list OR; the number selected is previewed.
- **By Group...** — selects the members of a group created in **Definition** → **Groups** (M5.1).

M3.2 Draw

- **Draw Frame** — click points in sequence in 2D or 3D; points snap to nodes, frame midpoints, then grid intersections. Right-click or Esc ends the chain (the command stays active), a second Esc leaves it. The section of new frames is chosen in the **Section** dropdown. Points that overlap on the 3D screen open a menu of coordinates to choose from.
- **Draw Area** — click 3 or 4 corners in one plane (triangle or convex quadrilateral) for a slab or wall; Enter / right-click after 3 points makes a triangle. Areas distribute loads to the frames on their edges (→ R2.8).
- **Quick Area** — in a Plan view, click inside a grid cell to create an area over it.
- **Node by Coord...** — a node at typed coordinates (e.g. a roof apex not on the grid).
- **Frame by Nodes...** — a frame from node i to node j chosen in the dialog.

A node lying in the middle of another frame is joined to it automatically at analysis (a secondary beam may rest at the middle of a girder without dividing the girder) → R1.6.

M3.3 Geometry: Node/Frame Tables

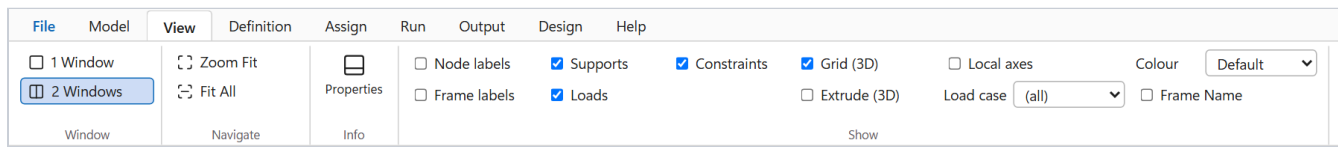
Opens the **Nodes** and **Frames** tabs in the bottom panel. Labels, node coordinates and the start/end nodes of frames can be edited; the last row adds a new object. A block of cells from a spreadsheet can be pasted (decimal commas are accepted; header rows are skipped). Changes apply on Enter or on leaving the cell; duplicate coordinates and duplicate connectivity are rejected with a message in the status bar.

M3.4 Modify

- **Move** — click a base point then a target point, or type dX/dY/dZ in the small panel above the view. Frame ends shared with unselected frames stay put (the selected frames get new nodes), so the other frames do not change.
- **Copy** — like Move but copies; copies keep the section, local axes, releases, offsets, tension/compression behaviour, design overwrites and frame loads.
- **Replicate...** — **Linear** (dX/dY/dZ × number of copies; a button picks two points to fill in the distance), **Radial** (rotate about an X/Y/Z axis by an angle per copy), **Mirror** (about the plane X/Y/Z = value or any plane), and **Move**. Global load directions and local axes are rotated/mirrored with the copy; supports stay on the global axes.

- **Divide** — divides frames by count, distance, intersections with other frames, or intersections with grid lines. Loads are divided with them; releases stay at the outer ends only.
- **Join** — merges collinear frames connected at a node without another frame, support, constraint or load; section and local axes must be the same.
- **Delete** (Del) and **Remove Free Nodes** (delete unused nodes).

M4. View menu



View tab.

- **1 Window / 2 Windows** — number of views. The view type of each window is chosen in its header.
- **Zoom Fit (F) / Fit All** — fit the model in the active window / all windows.
- **Properties** — opens the Properties panel for the selected object (it follows the selection).
- **Show: Node labels, Frame labels, Supports, Loads** (with a **Load case** filter; symbol size proportional to the load value), **Constraints** (purple markers), **Grid (3D)**, **Extrude (3D)** (true section shapes drawn in 3D), **Local axes** (arrows 1 red, 2 green, 3 blue — → **R1.2**).
- **Colour** — one colour or one colour per section with a legend. **Frame Name** writes the section id (S1, S2, ...) on each frame.

In a **Plan** view, columns connected to that level are drawn as their sections: continuous (filled), ending at this level (dashed outline), or starting at this level (filled + cross; a transfer column when it stands on a beam).

M5. Definition menu

File	Model	View	Definition	Assign	Run	Output	Design	Help		
										
Grid	Groups	Materials	Frame Sections	Shell Sections	Constraints	Load Cases	Load Combos	Combo Groups	Seismic	Mass Source
Geometry			Properties		Joint		Loads			Mass

Definition tab.

M5.1 Geometry: Grid and Groups

- **Grid** — coordinates of the X and Y grid lines and the Z levels, and the **Z level names** (Base/Story1..., prefix + number, elevation value, or manual). Level names are used in the grid bubbles, the plane lists and *By Property*.
- **Groups** — named sets of nodes and frames: **New** from the current selection, replace/add/remove members, rename, copy, delete. Groups do not affect the analysis.

M5.2 Properties: Materials, Frame Sections, Shell Sections

- **Materials** — E, G, mass density and strength (F_y/F_u for steel, f_c' for concrete). **Fill from preset:** steel BJ 34–55 and ASTM, concrete f_c' 20–40 MPa or grade K ($f_c' = 0.83 \cdot K/10$). Self weight = mass density $\times A \times g$ ([→ R1.1](#), [R2.3](#)).
- **Frame Sections** — rectangular concrete $b \times h$ or circular D; steel WH, I-Beam, UNP, CNP, angle, SHS, RHS, pipe and CT from the JIS database or custom sizes (lipped channel and lipped Z: custom), with a section drawing and automatic properties (A, I33, I22, J [→ R1.4](#)). **Property modifier:** factors on A, I33, I22, J, mass, weight; buttons *Cracked beam 0.35I_g / Cracked column 0.70I_g* ([→ R1.5](#)). A concrete column section can be given **provided bars** (diameter, number of bars per side or around the perimeter, spiral) so that the column is checked with those bars (*check mode* [→ R7.3](#)).
- **Shell Sections** — thickness and material of slabs/walls for the self weight of areas.

M5.3 Joint: Constraints

Joint constraint definitions: **Rigid**, **Diaphragm** (slab rigid in its plane), **Plate**, **Beam**, or **Custom** (choose DOFs), the normal axis, and the option of one constraint per level. Assigned through **Assign** [→ Joint/Area](#) [→ Constraints](#). Theory [→ R1.8](#).

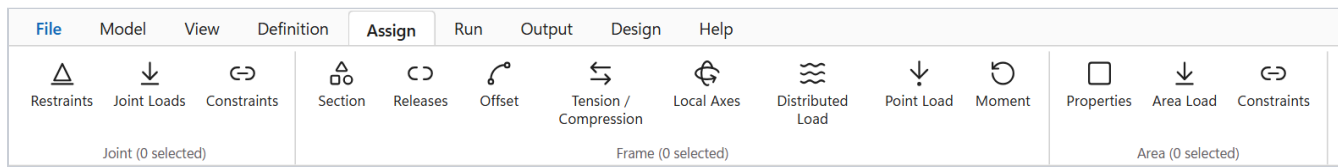
M5.4 Loads: Load Cases, Load Combos, Combo Groups, Seismic

- **Load Cases** — name, **type** (Dead, Superdead, Live, Roof live, Rain, Wind, Quake, Notional, piping, equipment, operating/test contents, thermal and others [→ R2.4](#)) and **self-weight factor** (default 1 only in the first Dead case). The **Notional** type creates SNI 1729 notional loads from a base case (**Notional DAM** button [→ R2.5](#)). The **Quake** type has a **Lateral load** row: *Manual*, *Auto (ELF)*, *Semi-auto (C_s, k)* or *Response spectrum (CQC/SRSS)* with direction X/Y ([→ R3](#)).
- **Load Combos** — list of combinations {load case $\times$ factor}, type *Strength* (LRFD) or *Service* (ASD), and a description. **Combination generator:** basis SNI 1727:2020 + SNI 1726:2019 or PIP STC01015 Tables 4/5, strength/service type, S_{DS} , ρ , wind & earthquake in both directions ($\pm$), earthquake 100 % + 30 %, L factor = 0.5, notional loads in gravity combinations. Name format: 1.4DEAD or 1.4D. Combination details [→ R2.7](#).
- **Combo Groups** — named groups of combinations. Envelopes can be limited to a group (*Env · group name*); a group marked **design** sets the combinations used by Check Design ([→ R2.9](#)).
- **Seismic** — SNI 1726:2019 parameters: S_s , S_1 , site class SA–SE (F_a/F_v automatic or manual), TL, R, I_e, structure type for T_a. S_{DS} , S_{D1} , T₀, T_s and the spectrum graph are shown immediately ([→ R3.1](#)).

M5.5 Mass: Mass Source

Source of mass for modal analysis and earthquake loads: element self mass (from mass density) and/or mass from load cases $\times$ factor (e.g. SDL $\times$ 1, LIVE $\times$ 0.25). Full rules [→ R2.6](#).

M6. Assign menu



Assign tab. Group titles show the number of selected objects.

Assign commands act on the current selection. Most dialogs have **Add**, **Replace** and **Delete** modes per load case. Every assigned value can also be seen and edited in the **Properties** panel ([M10.3](#)).

M6.1 Joint

- **Restraints** — *Assign Joint Restraints & Springs*: restraint per DOF (UX, UY, UZ, RX, RY, RZ; presets fixed, pinned, roller) and **support spring** k per DOF (0 = no spring; a restrained DOF disables its spring). → [R1.9](#).
- **Joint Loads** — forces and moments at nodes (global axes) per load case.
- **Constraints** — assigns/removes a constraint ([M5.3](#)) on the selected nodes; conflicting DOFs are rejected. Constrained nodes are marked with a purple diamond.

M6.2 Frame

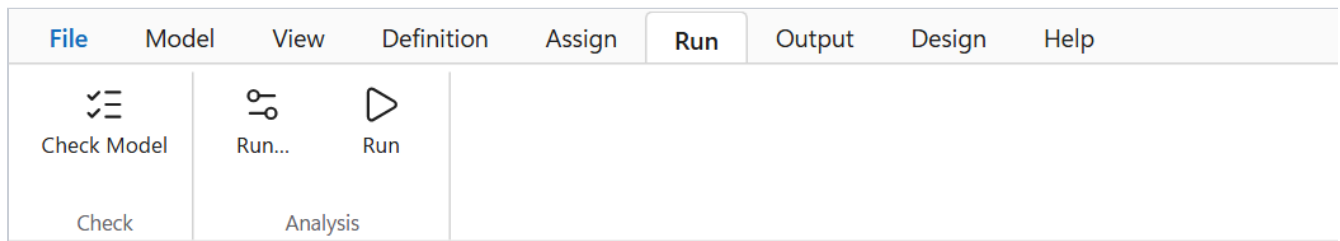
Command	Content	Theory
Section	Section of the selected frames	R1.4
Releases	Releases P, V2, V3, T, M2, M3 at end i/j and partial moment release factors M2/M3 (0 = fixed, 1 = pinned); presets pinned both ends, truss, etc. Unstable combinations are rejected at once	R1.7
Offset	Rigid end offsets (global $\Delta X/\Delta Y/\Delta Z$ or local $\Delta 1/\Delta 2/\Delta 3$ per end) and insertion point (cardinal point 1–10, <i>Transform</i> stiffness option)	R1.10
Tension / Compression	<i>Normal, Tension only, Compression only</i> (limit 0 kN); combinations become non-linear	R5.4
Local Axes	Rotation of axes 2 & 3 about axis 1 (set or rotate by; buttons 0/90/180/–90°)	R1.2
Distributed Load	Uniform/partial/trapezoidal/triangular load (four position–value pairs, relative or absolute); directions Gravity, Global X/Y/Z, Local 1/2/3; <i>per projected length</i> option	R2.1 , R2.2
Point Load	Point loads at several distances along the frame	R2.1
Moment	Concentrated moments about local axis 1/2/3 or uniform torsion along the frame	R2.1

Markers in the views: released ends = teal circles (partial = filled dot), tension/compression-only members = letters T/C, offset arms = thick lines.

M6.3 Area

- **Properties** — shell section (for self weight) and **two-way** or **one-way** distribution with span direction X/Y/Z. → [R2.8](#).
- **Area Load** — uniform load per area (gravity or normal to the plane) per load case; distributed to the edge frames.
- **Constraints** — a constraint (e.g. a diaphragm) for every node of the area: corners, nodes on the edges and nodes inside.

M7. Run menu



Run tab.

M7.1 Check Model

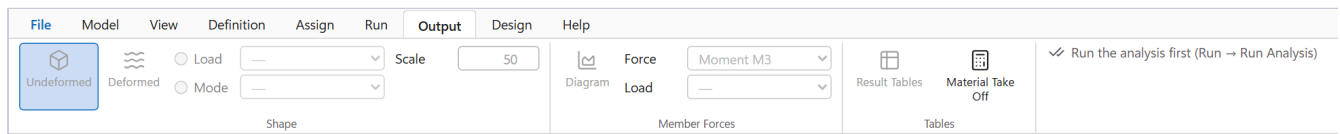
Checks the model without solving, in two stages: (1) in the browser — completeness of the model and assignments, coincident nodes, coordinates off the 0.1 mm grid; (2) on the server — builds the analysis model and tests the stiffness matrix; mechanisms are reported down to the node and DOF. The **Clean up** section offers **Merge Nodes** (default tolerance 0.001 m) and **Round Coordinates**. When a Run fails, Check Model opens automatically with the reason.

M7.2 Run... and Run

- **Run...** (Shift+F5) — options dialog:
 - **Static** — all load cases and combinations;
 - **P-Delta (second order) for load combinations** → [R5.1](#);
 - **Direct analysis method (SNI 1729 Chapter C)** — forces P-Delta, 0.8 steel stiffness and $0.001 \cdot Y_i$ notional loads → [R5.2](#);
 - **Modal (free vibration)** with **Number of modes** → [R4.3](#). The dialog shows an estimate of the server memory; models that are too large get a warning before they are sent.
- **Run** (F5) — runs at once with the last settings of Run... (default: static + modal, 6 modes).

Modal analysis runs automatically when there is an Auto or Response spectrum earthquake case. Combinations with tension/compression-only members are always analysed non-linearly. Meaning of each option combination → [R5.5](#).

M8. Output menu



Output tab.

M8.1 Shape

Undeformed / Deformed. Source of the deformed shape: **Load** (load case, combination or envelope) or **Mode** (vibration mode). The scale is computed automatically per case and can be changed. The deformed shape is drawn curved along the members. Selected frames get a **Uz max** deflection label, selected nodes get Ux/Uy/Uz.

M8.2 Member Forces

Diagram of member forces **N, V2, V3, T, M2, M3** per load case, combination or envelope (*Env · all, Env · strength, Env · group*). Blue fill = positive, red = negative; moments are drawn on the tension side. In 2D the end and span-extreme values are written; in 3D only for the selected frames. Envelopes are labelled with the governing combination. Sign convention → [R4.2](#).

M8.3 Tables: Result Tables and Material Take Off

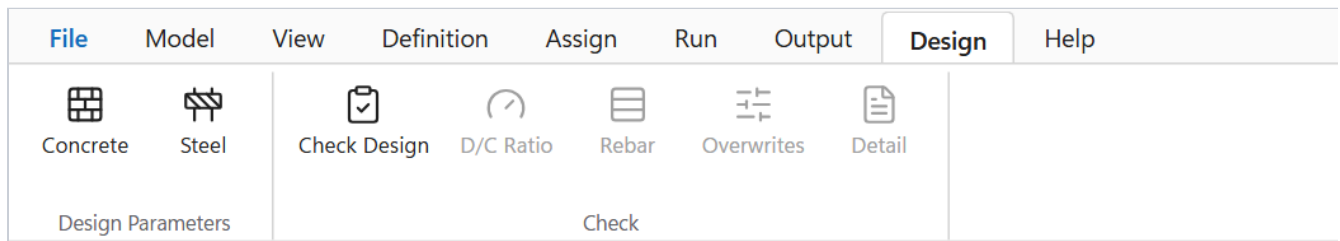
Result Tables opens the result tables in the bottom panel:

Tab	Content
Displacement	Node displacements, or displacements of the selected frames per station including Uz max between stations
Reactions	Support and spring reactions
Member Forces	Forces per station (selected frames, max. 200 frames); two rows at concentrated loads
Modal Periods	Periods, frequencies, mass participation ratios UX/UY/UZ and their cumulative sums
Seismic	W, Ta, Cu·Ta, modal T, T used, Cs, k, Vt, scale factors, V per earthquake case (→ R3)
Diaphragms	Centre of mass and centre of rigidity of each diaphragm (→ R4.4)
Steel Design / Concrete Design	Design results (M9)

The table header shows the analysis type (second order, amplification, iterations). Every tab can be exported to **.xlsx** (all rows, including the governing combinations of envelopes).

Material Take Off — length/area, volume, weight and mass per section and material; needs no analysis; can be exported to **.xlsx**.

M9. Design menu



Design tab.

M9.1 Design Parameters

- **Concrete** (SNI 2847:2019) — frame system OMF/IMF/SMF (SRPMB/SRPMM/SRPMK), ϕ factors, f_y/f_{yt} , cover, bar and stirrup diameters, beam design at the column face, *Ignore beam torsion*. Defaults → [R7.1](#).
- **Steel** (SNI 1729:2020) — **LRFD** or **ASD** method, ϕ factors, global K, ratio limit. Defaults → [R6.2](#).

Parameters are stored in the model and do not clear the analysis results.

M9.2 Check Design

Designs every frame from the last Run, using the strength combinations (or a group marked design):

- **Steel** — D/C ratio per frame (check mode); frames coloured by ratio with a legend; **Steel Design** tab. Strength per section type → [R6.3](#).
- **Concrete** — required reinforcement and provided bars (e.g. $6D19 / 3D19 \cdot \emptyset 10-100$), A_v/s , torsion, column ratio; **Concrete Design** tab. → [R7](#).

Warnings appear when the run does not fit (e.g. direct analysis without notional loads, slender sway columns without P-Delta, concrete without cracked modifiers in a second-order run).

M9.3 Display and Detail

- **D/C Ratio** — turns the ratio colours on/off.
- **Rebar** — reinforcement labels on concrete frames (beams: top/bottom/stirrups at support–span–support; columns: bars and ties) instead of ratio labels.
- **Overwrites** — per selected frame: steel $L_x/L_y/L_b$ (ratios), K, C_b , angle in a planar/space truss and connected leg, stiffener spacing and stiffener plates, J10 concentrated forces; concrete *Design as* beam/column, slender column non-sway / sway, and column k. → [R6.2](#), [R7.3](#).
- **Detail** — step-by-step calculation report for the selected frame: clauses, formulas, values, ratios; for concrete also the zone table, sections, stirrup elevation and the column P-M interaction diagram (→ [R7.4](#)).

M10. Help menu, navigation and panels

File	Model	View	Definition	Assign	Run	Output	Design	Help
 Units	 Shortcuts	 About	 Legal					
Settings	Help	Information						

Help tab.

M10.1 Help menu

- **Units** — change the model units ([M1.5](#)).
- **Shortcuts** — list of keyboard and mouse shortcuts ([M10.2](#)).
- **About** — application version and release notes.
- **Legal** — Terms of Service, Privacy Policy, Technical Limitations, verification statement, third-party licences (in Indonesian).

M10.2 Shortcuts and mouse

Key / gesture	Command
Ctrl+N / Ctrl+S	New / Save
Ctrl+Z / Ctrl+Y (Ctrl+Shift+Z)	Undo / Redo
Ctrl+A / Delete	Select all / delete the selection
Space	Select mode from any command (the selection stays)
Esc	Finish drawing → leave the command → clear the selection
F / F5 / Shift+F5	Zoom fit / Run / Run...
Shift or Ctrl + click	Add to the selection
Double-click node/frame	Select + open Properties
2D: scroll / middle or right drag	Zoom at the cursor / pan
3D: left drag (Select)	Selection box (left→right <i>window</i> , right→left <i>crossing</i>)
3D: right drag, Shift + drag, Alt + left drag	Orbit (with drawing commands: left drag also orbits)
3D: middle drag / scroll	Pan / zoom

M10.3 Properties panel and tables

The **Properties** panel shows the selected object and can be edited directly: node coordinates and supports, frame section and local axes, and **every assigned load** (value, direction, position, load case, delete per row). The frame card also shows A, I, J and the max/min member forces for the load chosen in the Output tab. Entries apply on Enter or on leaving the field; Esc cancels; everything can be undone.

M11. Example: 3D concrete frame

This example uses the built-in template and touches the main steps. The numbers are only illustrative.

1. **File → New:** kN, m; grid X 4 lines × 6 m, Y 3 lines × 5 m, Z 4 levels × 3.5 m; template *3D frame on the grid*.
2. **Definition → Materials:** concrete preset $f_c' = 25$ MPa.
3. **Definition → Frame Sections:** S1 = B30x50 (rectangle 300 × 500 mm, *Cracked beam* modifier), S2 = K50x50 (*Cracked column* modifier).
4. **Definition → Load Cases:** DEAD (Dead, self weight 1), SDL (Superdead), LIVE (Live), EX and EY (Quake, *Auto (ELF)* direction X/Y). **Definition → Seismic:** enter S_s , S1, site class, $R = 8$, $I_e = 1$.
5. **Definition → Mass Source:** self mass + $SDL \times 1 + LIVE \times 0.25$.
6. **Model → By Property...:** select all beams; **Assign → Frame → Distributed Load:** SDL 10 kN/m, LIVE 5 kN/m (Gravity).
7. Select all floor nodes; **Assign → Joint → Constraints:** Diaphragm, one per level.
8. **Definition → Load Combos → generator:** SNI, strength, S_{DS} from Seismic, $\rho = 1.3$, earthquake 100 % + 30 %.
9. **Run → Check Model**, then **Run...:** Static + P-Delta + Modal with 12 modes.
10. **Output → Deformed** (mode 1), **Member Forces → Diagram** M3 envelope; **Result Tables → Seismic** to check T and Vt.
11. **Design → Concrete:** SMF; **Check Design; Rebar** for reinforcement labels; select a column → **Detail**.
12. **File → Report → Full report**, then **File → Save**.

Part B — Technical Reference

R1. Model and elements

R1.1 Units

- The solver works in **consistent units**. Model units are chosen at *New* (length m or mm, force N or kN) and can be changed through **Help → Settings → Units**: every model value is converted (**M1.5**).
- Gravity $g = 9.80665 \text{ m/s}^2$, expressed in the model length unit (e.g. 9806.65 mm/s^2).
- The material density is a consistent **mass** per volume ($\text{force} \cdot \text{s}^2 / \text{length}^4$, e.g. t/m^3 for kN-m). Self weight = density $\times$ A $\times$ g.
- Design calculations work in MPa and mm; material strengths (f_y , f_u , f_c') are always entered in MPa.

R1.2 Axis systems

Global: right-handed X-Y-Z, **Z up**, gravity towards $-Z$.

Frame local axes (axes 1-2-3, the usual convention of 3D frame analysis programs):

Axis	Direction
1	Along the member, from node i to node j
2	Beams/inclined members: "up" (projection of +Z perpendicular to the member). Vertical columns: global +X
3	1 $\times$ 2 (right hand)

- Rotating the local axes (**Assign → Frame → Local Axes**) turns axis 2 towards axis 3 about axis 1. A beam along X rotated by $90^\circ \rightarrow$ axis 2 = $-Y$, axis 3 = $-Z$.
- Vertical bending of beams is governed by **I33** (strong axis of the section). The strong axis of section tables (I_x) maps to I33, the weak axis (I_y) to I22.

R1.3 3D frame element

- **3D Euler-Bernoulli element, 2 nodes, 12 DOF** (3 translations + 3 rotations per node). Classical local stiffness matrix: axial EA/L , torsion GJ/L , bending in two planes ($12EI/L^3$, $6EI/L^2$, $4EI/L$, $2EI/L$), transformed to global with the local-axis rotation matrix.
- **Shear deformation is not included** (no shear area / Timoshenko). For deep beams or short spans, deflections are slightly smaller than in programs that include shear deformation by default.
- Cubic Hermite shape functions: point loads and linear distributed loads are converted to consistent equivalent nodal loads, so static results at the nodes are **exact**.
- G is entered directly in the material (not computed from Poisson's ratio).
- No warping stiffness (C_w) in torsion. Rigid end offsets through end offsets / insertion points (**R1.10**); a *rigid-zone factor* and automatic offsets from column width are deliberately not provided.

R1.4 Section properties

- The solver reads **A**, **I22**, **I33**, **J** (and the density for weight and mass).
- Steel sections from the JIS database (WH, I-Beam, UNP, CNP, angle, SHS, RHS, pipe; 399 sizes) were copied from the official tables and cross-checked against their geometry. CT is made from half a WH. Custom sizes: properties are computed from the section polygon.
- **J is not in the JIS tables** and is always computed: open sections $\Sigma b \cdot t^3 / 3$ on the centre line, SHS/RHS Bredt's formula (centre-line radius $1.5t$), pipes exact.
- **Angles and Z sections** are analysed on their geometric axes (I_{xy} neglected) unless the **principal axes** option is chosen in the section: the stiffness is then rotated to the principal axes u/v . I_u , I_v and $\tan \alpha$ are used in design.

- Concrete: rectangle $b \times h$ and circle D . "General" sections: properties entered directly.

R1.5 Property modifiers

Purpose: an effective stiffness different from the gross section — mainly **cracked concrete sections** (SNI 2847 § 6.6.3.1.1) — without changing dimensions.

- Factors per section: **A, I22, I33, J, mass, weight** (default 1). Preset buttons: *Cracked beam 0.35I_g, Cracked column 0.70I_g*.
- Applied in every solver path: static stiffness, modal, P-Delta (including the axial force that drives K_g) and the recovery of released ends. Mass and weight are separate from the A factor.
- **Design always uses the gross section** (modifiers are for analysis only).
- Per section only; beams and columns of the same size need two sections.
- **One set for all analyses:** the service stiffness of $1.4 \times$ the cracked value (§ 6.6.3.2.2) and the division by $(1 + \beta ds)$ for sustained lateral loads are not applied (**R5.3**).

R1.6 Analysis model (discretisation)

The solver does not compute directly from the drawn frames. At Run an **analysis model** is created:

1. **Translate** — high-level loads, area loads and self weight are converted to primitive loads.
2. **Join nodes inside frames** — a model node lying on the line of a frame (between its ends) is connected to that frame, so a secondary beam may rest at the middle of a girder without dividing the girder. Two frames crossing without a node are **not** connected.
3. **Discretise** — every frame is divided into **6 collinear sub-elements**, plus cut points under point loads/moments and at the start/end of partial distributed loads. Division points within $\frac{1}{4}$ of a division of a load are moved to the load position.
4. **Map loads** — loads to analysis nodes/sub-elements, per load case.

Reasons: results (deflections, member forces) are available at points along the member; the mass distribution is fine enough for modal analysis (cantilever period $\approx 1.3\%$ from the continuous solution with 6 sub-elements); and **P-δ** (member curvature) is captured in second-order analysis. Internal nodes are invisible; results are mapped back to the physical nodes and stations.

R1.7 End releases

- Releases per end in local axes: P, V2, V3, T, M2, M3.
- Method: **static condensation** of the end sub-element: $k^* = k_{aa} - k_{ar} \cdot k_{rr}^{-1} \cdot k_{ra}$; fixed-end loads are condensed the same way. The released force = 0 at that end; the joint itself stays intact.
- **Partial moment release** (factor $f = 0.001-1$) on M2/M3: a rotational spring in series between the joint and the member end, $k = (1 - f)/f \cdot 4EI/L$ (L = flexible length of the physical member, EI with modifiers). End rotational stiffness (far end fixed) = $(1 - f) \cdot 4EI/L$. $f = 1 \rightarrow$ full hinge; $f \rightarrow 0 \rightarrow$ fixed. With spring S : $M = k_{rr} + S$; $k^*_{aa} = k_{aa} - k_{ar} M^{-1} k_{ra}$, $k^*_{ar} = k_{ar} M^{-1} S$, $k^*_{rr} = S - S M^{-1} S$ ($S = 0$ gives the full release). Used identically in static, modal, P-Delta and force recovery. The end moment does not drop by f — a fixed-fixed beam under uniform load with the same f at both ends: $M = wL^2/12 \cdot 2(1 - f)/(2 - f)$ ($f = 0.5 \rightarrow \frac{2}{3} \cdot wL^2/12$).
- The rotation of a released member end is recovered for display (the deformed shape kinks at the hinge).
- Release combinations that make a member unstable are rejected: P, T, V2 or V3 at both ends; M3 at both ends + V2; M2 at both ends + V3.
- A joint where every frame releases a rotation (e.g. a truss) gets a very small stabilising spring ($10^{-10} \times$ the largest rotational stiffness) on that DOF only. If a moment load goes into that spring the analysis is rejected ("Mechanism ...").

R1.8 Joint constraints

Purpose: force a group of joints to move as a rigid body in selected DOFs (floor diaphragms, etc.).

Type	Tied DOFs
Rigid (body)	All 6
Diaphragm, normal Z	UX, UY, RZ (out-of-plane bending free)
Plate/wall, normal X / Y	X: UY, UZ, RX · Y: UX, UZ, RY
Beam (line), along an axis	All except rotation about that axis
Custom	Chosen DOFs

- **No master joint:** a virtual reference point at the centroid of the group; results do not depend on its location.
- Kinematics: $u_x = U_x + r_y \cdot d_z - r_z \cdot d_y$, $u_y = U_y + r_z \cdot d_x - r_x \cdot d_z$, $u_z = U_z + r_x \cdot d_y - r_y \cdot d_x$.
- Exact implementation through the transformation $u = T \cdot u_r$ ($K_r = T^T K T$, $F_r = T^T F$, $M_r = T^T M T$) — **not penalty springs**.
- "One per level" creates one diaphragm per elevation from a single assignment. Area constraints apply to every node of the area.
- DOFs held by supports are not tied (the restraint wins). One DOF of one joint belongs to one constraint only.

R1.9 Supports and support springs

- Restraint per DOF (6 DOFs per node): fixed, pinned, roller or any combination.
- **Support springs:** translational (force/length) and rotational (force-length/rad) stiffness per DOF in global axes, added to the diagonal of K — included in static, modal, P-Delta, tension/compression, direct analysis and constraints. Spring reaction = $-k \cdot u$. A restrained DOF ignores its spring (the restraint wins). Structures held only by springs are accepted.
- Inclined supports (local support axes) and compression-only supports (uplifting footings) are not available.
- A model without any support is rejected.

R1.10 End offsets and insertion points

Purpose: members whose axis does not pass through the node — beams from the column face, beams flush with the top of the slab, edge columns flush with the façade, eccentric connections — without extra nodes and short members.

Concept: rigid arms. The flexible member spans between end points E_i and E_j ; each point is tied to its node by a massless rigid arm: $u_E = u_P + \theta_P \times (E - P)$, $\theta_E = \theta_P$.

Input	Source	Changes stiffness
End offset	Node → member-end vector per end; global ($\Delta X, \Delta Y, \Delta Z$) or local ($\Delta 1, \Delta 2, \Delta 3$ on the axes of the node line)	Always
Insertion point	Cardinal point 1–10 on the bounding box of the section	Only when Transform is ticked (default off)

Cardinal points are viewed along +1 (from end i) with axis 2 up and axis 3 to the right: 1–3 bottom, 4–6 middle, 7–9 top; 1/4/7 left (–3), 3/6/9 right (+3); 10 = centroid. "Middle" = middle of the bounding box (UNP/angle: 5 ≠ 10).

Geometry: $Q = P + \text{offset}$; local axes from the line $Q_i \rightarrow Q_j$; the cardinal point (c_2, c_3) is placed on that line, so the centroidal axis = $Q - (c_2 \cdot l_2 + c_3 \cdot l_3)$. Offsets that use up or reverse the member length are rejected.

Method: E becomes an analysis node eliminated exactly through $u = T \cdot u_r$ (the same mechanism as constraints [R1.8](#), without penalty springs and without extra DOFs). When the node is also constrained, $T = T_{\text{link}} \cdot T_{\text{constraint}}$.
Reactions = $L^T(K \cdot u - F)$, so forces at the arm end go to the support. Elements, releases, P-Delta, tension/compression-only, modifiers and loads act on the flexible member as usual.

Rules to know

- **Releases** act at the end of the flexible member (at E, the support face).
- **Loads:** frame load positions are measured from E_i along the flexible member; a "full" distributed load covers the flexible part. Rigid arms carry no load, weight or mass. A beam offset to the column faces loses $w \cdot a$ per end — in buildings that load is normally carried by the column anyway.
- **Output:** stations from E_i to E_j; end moments = moments **at the support face**.
- **Why transformed insertion points are off by default:** without shell slabs, a transformed *top-center* beam in a rigid-diaphragm floor behaves like a composite beam with an infinitely stiff slab — under gravity, tension appears in the beam and the span moment and deflection drop (test frame: -13 % and -19 %), making the beam design unconservative. Pure lateral drift hardly changes.
- **Limitation:** second-order effects of the rigid arms themselves are neglected (K_g only from the flexible part).

R2. Loads and combinations

R2.1 Frame loads

Type	Parameters	Processing
Distributed (full / partial / trapezoidal / triangular)	Position–value pairs (relative 0–1 or absolute)	The member is cut at the start & end of the load; each sub-element receives its linear load as consistent equivalent nodal loads (exact for linear loads)
Point	P, distance	The member is cut exactly under the load; the station is written twice (left & right) so the shear jump is drawn
Concentrated moment	M about local axis 1/2/3, distance	Like a point load (nodal moment at the cut point)
Uniform torsion	m per length, full span	$m \cdot L/2$ at each end of the sub-element (exact)
Joint load	Force & moment at a node	Directly into the load vector

Uniform moments about axes 2/3 are deliberately not provided. On offset frames, load positions are measured from the offset end i and full-span loads cover only the flexible part ([R1.10](#)); self weight also covers only the flexible part.

Frame load directions: *Gravity* (default) –Z, positive values downwards; *Global X/Y/Z* positive along the axis; *Local 1/2/3* follow the member local axes (beam along X: 1 = +X, 2 = +Z, 3 = –Y). Distributed loads are stored as global vectors per sub-element and resolved into local axes at assembly — **exact for columns and inclined members**.

R2.2 Loads per projected length

- For loads defined per plan/projected area (e.g. sloped roofs), directions Gravity / X / Y / Z only.
- Intensity along the member = $w \cdot \sin(\text{member-load angle}) = w \cdot \sqrt{1 - (\hat{x} \cdot d)^2}$.
- Example: a 4 : 3 sloped roof with projected gravity carries $w \times 4$ (plan length); a column carries 0. Projected X load carries $w \times \text{height}$.

R2.3 Self weight

- Self weight = density $\times$ A $\times$ g per length (–Z), multiplied by the **self-weight factor** of each load case.
- Default: factor 1 only for the **first Dead load case**, 0 for the others — so self weight is not counted twice. The "weight" factor of a property modifier multiplies the self weight of that section.
- Load-distributing areas: density $\times$ thickness $\times$ g per area, with the same factor ([R2.8](#)).

R2.4 Load case types

D dead, SD superdead, L live, Lr roof live, R rain, W wind, Wp partial wind (during test, PIP cl. 4.1.4.5), E earthquake, F fluid, H soil pressure, CR crane/lifting, P piping, EQ equipment (empty weight), OP operating contents, HT test medium (hydrotest), Ts sustained thermal, Tt temporary thermal, O other, N notional. Thermal types are entered loads (e.g. pipe anchor forces = Ts, pipe friction at start-up = Tt); temperature loads $\alpha \cdot \Delta T$ on frames are not available. The solver only distinguishes **dead** (default self weight), **notional**, and **quake** with generated loads ([R3](#)); the other types are used by the combination generator.

R2.5 Notional loads (SNI 1729 § C2.2b)

Purpose: to represent initial geometric imperfections (out-of-plumbness) for the direct analysis method.

- A load case of type **notional**: {base case, ratio (default 0.002), direction X or Y}. The **Notional DAM** button creates N...X and N...Y for every gravity case (dead, superdead, live, piping, equipment, crane).

- At **every node of the analysis model** (including internal sub-element nodes): lateral force = ratio × the downward vertical load of the base case at that node (including self weight). Nodes with a net upward load get no notional load.
- A notional case cannot carry other loads and has no self weight. Directions -X / -Y through negative factors in combinations.
- The additional notional load $0.001 \cdot Y_i$ ($\tau_b = 1$) is **not** a load case: it is added automatically when the direct analysis method is active (R5.2).

R2.6 Mass source

- **Self mass** (on by default): density × A × L, half to each end of a sub-element (the mass modifier applies); load-distributing areas: density × thickness × area.
- **Mass from load cases × factor**: load components entered **downwards** (-Z) are converted to mass W/g, acting in translations X, Y, Z. Upward loads, X/Y loads and moments add no mass.
- Self weight produced by load-case factors is **not** taken through the load-case path, so self mass is not counted twice.
- **Linear loads** (triangular/trapezoidal, e.g. area loads transferred to beams) are lumped per sub-element with **simple-beam reactions**: $h \cdot (2w_i + w_j)/6$ and $h \cdot (w_i + 2w_j)/6$. Some programs use fixed-end beam shears $(3/20 \cdot 7/20)$; the total mass is the same and periods differ very little (± 0.05 % on the test model).
- Zero total mass → modal analysis is rejected.

R2.7 Load combinations

- A combination = a list of {load case: factor}. Linear analysis: combination results = **superposition** of load-case results. Non-linear analysis (P-Delta, direct analysis method, tension/compression-only): each combination is **analysed on its own**.
- Each combination is of type **strength** (LRFD, default) or **service** (ASD) and has a free description (the generator fills in the clause number). Neither is read by the solver. Check Design uses strength combinations (ASD steel: service) unless a group marked design says otherwise.

Strength (LRFD) combination generator — SNI 1726:2019 cl. 4.2.2.1 (= SNI 1727:2020 cl. 2.3.1) + cl. 7.4.2.3:

No.	Factors
1	1.4D (+ 1.4F, 1.6H)
2	1.2D + 1.6L + 0.5(Lr or R) (+ 1.6CR, 1.2F, 1.6H)
3	1.2D + 1.6(Lr or R) + (L or 0.5W) — only with Lr/R cases
4	1.2D + 1.0W + L + 0.5(Lr or R) (L = 1.0 or 0.5)
5	$(1.2 + 0.2 \cdot S_{DS})D + \rho E + L$
6	0.9D + 1.0W
7	$(0.9 - 0.2 \cdot S_{DS})D + \rho E$

Service (ASD) combination generator — SNI 1726:2019 cl. 4.2.3.1 (= SNI 1727:2020 cl. 2.4.1) + cl. 7.4.2.3; F and H factor 1.0:

No.	Factors
1	D
2	D + L
3	D + (Lr or R) — with Lr/R cases
4	D + 0.75L + 0.75(Lr or R) — with Lr/R cases

No.	Factors
5	$D + 0.6W; (1.0 + 0.14 \cdot S_{DS})D + 0.7pE$
6a	$D + 0.75L + 0.75(0.6W) + 0.75(Lr \text{ or } R)$
6b	$(1.0 + 0.105 \cdot S_{DS})D + 0.75L + 0.75(0.7pE) + 0.75(Lr \text{ or } R)$
7	$0.6D + 0.6W$
8	$(0.6 - 0.14 \cdot S_{DS})D + 0.7pE$

"Lr or R": each Lr/R case is taken one at a time. **Thermal** (option): each Ts or Tt case one at a time with factor 1.0 (strength, SNI 1727 cl. 2.3.4) or 0.75 (service, cl. 2.4.4), alongside the combinations without thermal; not in uplift combinations. Thermal $\pm$ directions are made as two cases.

Generator options: S_{DS} (filled from Definition $\rightarrow$ Seismic when present), p (1.0 / 1.3), W and E in both directions ($\pm$), **100 % + 30 %** for two or more earthquake cases (SNI 1726 cl. 7.5.3), L factor 0.5 in strength W/E combinations, and notional loads (strength only: each gravity combination becomes +X, -X, +Y, -Y with notional factor = the factor of its base case; lateral combinations without notional loads, § C2.2b(4)). *Replace* only replaces combinations of the same type.

Equipment contents (SNI basis): piping, equipment and operating contents OP are treated as dead load; operating contents are left out of uplift combinations 6/7; test medium HT and partial wind Wp are not generated.

PIP STC01015 (April 2017) cl. 4.2.2.2, buildings & open frames — Table 5 (strength) and Table 4 (service). $D_s = D$ (dead + superdead); $D_e =$ piping + equipment (empty weight); $D_o = D_e +$ operating contents OP; $D_t = D_e +$ test medium HT (cl. 4.1.2.3). Without an OP case, piping + equipment is taken as D_o and the empty condition is D_s only (conservative for uplift).

No.	Table 5 (strength)	No.	Table 4 (service)
5-1.1	$1.4(D_s + D_o) + 1.2T_s$	4-1.1	$D_s + D_o + T_s$
5-1.2	$1.4(D_s + D_t) + 1.2T_s$	4-1.2	$D_s + D_o + T_s + T_t$
5-2.1	$1.2(D_s + D_o) + 1.2T_s + 1.0T_t + 1.6L + 0.5(Lr \text{ or } R)$	4-1.3	$D_s + D_t + T_s + T_t$
5-2.2	$1.2(D_s + D_t) + 1.2T_s + 1.0T_t + 1.6L + 0.5(Lr \text{ or } R)$	4-2.1 / 4-2.2	$D_s + D_o/D_t + T_s + L$
5-3.1 / 5-3.3	$1.2(D_s + D_o/D_t) + 1.2T_s + 1.0T_t + 1.6(Lr \text{ or } R) + 0.5L$	4-3.1 / 4-3.2	$D_s + D_o/D_t + T_s + (Lr \text{ or } R)$
5-3.2 / 5-3.4	$1.2(D_s + D_o/D_t) + 1.2T_s + 1.6(Lr \text{ or } R) + 0.5W$	4-4.1 / 4-4.2	$D_s + D_o/D_t + T_s + 0.75T_t + 0.75L + 0.75(Lr \text{ or } R)$
5-4.1	$1.2(D_s + D_o) + 1.2T_s + 1.0W + 0.5L + 0.5(Lr \text{ or } R)$	4-5.1	$D_s + D_o + T_s + 0.6W$
5-4.2	$1.2(D_s + D_t) + 1.2T_s + 1.0W_p + 0.5L + 0.5(Lr \text{ or } R)$	4-5.2	$D_s + D_o + T_s + 0.7E_o$
5-5	$1.2(D_s + D_o) + 1.2T_s + 1.0E_o + 0.5L$	4-5.3	$D_s + D_t + T_s + 0.6W_p$
5-6.1	$0.9(D_s + D_e) + 1.2T_s + 1.0W$	4-6a.1 / 4-6a.2	$D_s + D_o/D_t + T_s + 0.75L + 0.75(0.6W/W_p) + 0.75(Lr \text{ or } R)$
5-6.2	$0.9(D_s + D_o) + 1.2T_s + 1.0W$	4-6b	$D_s + D_o + T_s + 0.75L + 0.75(0.7E_o)$
5-7.1	$0.9(D_s + D_e) + 1.2T_s + 1.0E_e$	4-7.1 / 4-7.2	$0.6(D_s + D_e/D_o) + T_s + 0.6W$
5-7.2	$0.9(D_s + D_o) + 1.2T_s + 1.0E_o$	4-8.1 / 4-8.2	$0.6(D_s + D_e/D_o) + T_s + 0.7E_e/E_o$
5-8.1 / 5-8.2	$1.2(D_s + D_o/D_t) + 1.2T_s + 1.2T_t + 0.5L + 0.5(Lr \text{ or } R)$		

Application: Ts is in every combination, Tt only in the rows that name it (never together with W/E); several Ts/Tt cases are taken one at a time; 5-8.x and 4-1.2 only when a Tt case exists. The Ts factor in 5-1.x = 1.4 (note h); in uplift cases 1.2 / 1.0 (Ts assumed to add to uplift — edit manually when it resists). Test combinations (Dt, Wp) only when an HT case (and Wp) exists. Notes b/d/e of the PIP table are not applied automatically. L 0.5 (note f) through the L factor option; 5-8.x always 0.5L. Ee = Eo = the model's earthquake case (operating mass; conservative for 5-7.1 / 4-8.1); the vertical term $\pm 0.2 \cdot S_{DS} \cdot D$ is included. Snow S is not generated. F, H, crane/lifting follow the SNI rules.

Generator assumptions: F uses the dead-load factor (not in 0.9D + W / 0.6D + 0.6W); H is always 1.6 (strength) / 1.0 (service), assumed to add to the effect; crane/lifting follows the live-load factor (SNI 1727 cl. 4.9).

R2.8 Load-distributing areas

Purpose: floor slabs, roofs and walls that only pass loads to the beams/columns on their edges — without slab stiffness (the slab is assumed not to carry global bending; its in-plane stiffness is represented by a diaphragm constraint when needed).

- An area = a **convex, planar** polygon (≥ 3 corners, no three corners collinear), horizontal, sloped or vertical.
- **Area load** q per area of the plane: *gravity* ($-Z$, per area of the sloped plane) or *normal to the plane* (e.g. wind on a wall). Self weight of the area = density $\times$ thickness $\times$ g, with the load case's self-weight factor.
- **Two-way:** every point of the area is carried to the **nearest edge** (angle bisectors) $\rightarrow$ triangles/trapezoids on rectangular edges; square a: triangle with peak $q \cdot a/2$, total $q \cdot a^2/4$ per edge.
- **One-way** X / Y / Z (projected onto the plane of the area): the load spans parallel to that direction to the two edges it crosses; rectangle: uniform $q \cdot L/2$ on the edges perpendicular to the span.
- An area edge may consist of several frames (divided beams); a part of an edge without a frame passes its load to the nodes at the ends of the gap with simple-beam reactions (Check Model warns). Overlapping frames on one edge do not count the load twice.
- Σ distributed load = $q \times$ area (verified for triangles, general quadrilaterals, pentagons/hexagons, 3D sloped planes).
- Area mass is included in the mass source (self mass) and distributed with the same rule; mass from linear loads [R2.6](#).
- An area constraint applies to every node of the area (corners, nodes on the edges, nodes inside).
- Meshed plate elements (membrane/plate/shell) are **not available** in this version.

R2.9 Combination groups and envelopes

- **Combination group:** a named set of combinations. The **design** mark sets the combinations used by Check Design; without a group marked design, all strength combinations are used. Groups are not read by the solver.
- **Envelope:** max/min per node and per station over all combinations (or one group, or per strength/service type) for each component (displacement, reaction, N, V2, V3, T, M2, M3), with its governing combination. Computed from the same run without extra analysis.

R3. Earthquake loads to SNI 1726:2019

R3.1 Parameters and design spectrum

- **Definition → Seismic:** S_s , S_1 , site class SA–SE (F_a/F_v from Tables 6/7 with linear interpolation; manual F_a/F_v for SF or site-specific studies), T_L (Figure 20: 6/8/12/16/20 s, default 20), R , I_e , structure type for T_a (Table 18: Ct, x).
- $S_{MS} = F_a \cdot S_s$, $S_{M1} = F_v \cdot S_1$, $S_{DS} = \frac{2}{3} S_{MS}$, $S_{D1} = \frac{2}{3} S_{M1}$, $T_0 = 0.2 \cdot S_{D1} / S_{DS}$, $T_s = S_{D1} / S_{DS}$; spectrum § 6.4 (S_a rises linearly up to T_0 , flat S_{DS} up to T_s , S_{D1}/T up to T_L , $S_{D1} \cdot T_L / T^2$ beyond).
- Tables and formulas were copied from the text of SNI 1726:2019.

R3.2 Lateral load of a Quake load case

Option	Content
Manual	Loads assigned by the user
Auto (ELF)	Equivalent lateral force § 7.8: T = period of the mode with the largest mass participation in that direction, limited to $C_u \cdot T_a$ (§ 7.8.2); C_s eq. 31–35; k § 7.8.3; $F_x = C_{vx} \cdot V$ (eq. 40–41)
Semi-auto (Cs)	$V = C_s \cdot W$ with C_s and k from the user; same vertical distribution
Response spectrum	§ 7.9.1: $S_a(T_n) \cdot g \cdot I_e / R$ per mode, CQC (5 % damping) or SRSS

- Direction global +X or +Y; –X/–Y through negative combination factors. A generated earthquake case cannot carry assigned loads and has no self weight.
- **T for Cs:** $\min(\text{modal } T, C_u \cdot T_a)$ literally (§ 7.8.2, § 7.9.1.4.1); when modal $T < T_a$, modal T is still used (conservative).
- Modal analysis runs automatically when there are Auto/RS cases (number of modes from the Run dialog).

R3.3 Seismic weight and force distribution

- **W and w_x** from the Mass Source: masses of analysis-model nodes are lumped to the physical joints (lever rule along the member; offset ends through the rigid arm). W = mass above the base $\times g$; base = the lowest Z of supported nodes.
- ELF lateral forces are applied at the joints of each level according to $w_x \cdot h_x^k$.

R3.4 Response spectrum

- Forces and reactions are scaled to 100 % of the equivalent lateral force V when $V_t < V$ (§ 7.9.1.4.1); drifts are scaled to $C_s \cdot W / V_t$ **only when Cs is governed by eq. 35** ($S_1 \geq 0.6$ and the limit $0.5 \cdot S_1 / (R/I_e)$ governs).
- **Signed results:** CQC/SRSS values take the sign of the dominant mode term (the mode with the largest mass participation in the earthquake direction). An RS case then behaves like an ordinary load case: a $\pm E$ combination gives $S \pm |R|$; envelopes and design work as usual. Correlation between quantities (e.g. column P and M) follows the dominant mode.
- In non-linear combinations (P-Delta, direct analysis method, tension/compression-only) the RS case is added linearly on top of the non-linear solution of the other cases; the state of tension-only members and the $0.001 \cdot Y_i$ notional loads do not see the RS part. ELF cases are ordinary loads and take part fully.
- The **Seismic** tab warns when the mass participation is below 90 % (§ 7.9.1.1).

R3.5 Not yet available

Accidental torsion 5 % (§ 7.8.4.2), storey drift check (C_d/I_e , § 7.8.6) and the stability coefficient θ , seismic design category and automatic ρ , modal analysis with P-Delta stiffness. These checks are done by the user outside the application.

R4. Linear analysis

R4.1 Linear static analysis

Purpose: displacements, reactions and member forces from each load case and combination, with linear elastic material and small displacements (first order).

Direct stiffness method:

1. Assemble the global stiffness matrix K from all sub-elements (with property modifiers, releases, constraints, springs).
2. Build the load vector F for **every load case**.
3. Solve $K_{ff} \cdot u = F_f$ on the free DOFs. K_{ff} is factorised **once** (sparse LU), then all load cases are solved by back substitution.
4. Combination = **linear superposition** of load-case results.

Reactions: $R = K \cdot u - F$ at restrained DOFs. The $-F$ term makes the part of fixed-end loads that falls directly on a support appear as a reaction.

Detection of unstable structures: mechanisms (missing supports, joints without rotational stiffness loaded by a moment) are reported with the DOF location. Results must also satisfy their own equations: if $|K \cdot u - F| > 10^{-3} \times$ the largest load in a load case, the structure is declared unstable.

R4.2 Member forces and sign convention

- Six components per station, local axes: **N, V2, V3, T, M2, M3**.
- From the end displacements of each sub-element: $p = k \cdot u - f$ (f = fixed-end loads of the sub-element) — **exact** at the stations for point loads and linear distributed loads.
- Stations: sub-element ends (6 per frame + load cut points). Under point loads/moments there are two values (left & right). Between stations the diagram interpolates V linearly and M parabolically ($dM3/dx = -V2$, $dM2/dx = -V3$).

Sign convention — forces on the positive cut face (outward normal +axis 1):

Component	Positive
N	Tension
V2, V3	Along +2 / +3 on the positive face
T	Moment vector +1 on the positive face
M2	Tension on the -3 face
M3	Tension on the -2 face → beam span (sagging) moment positive

Moment diagrams are drawn on the tension side. Example: pin-roller beam with uniform load: $V2 = -wL/2$ at i , $+wL/2$ at j ; $M3 = +wL^2/8$ at midspan.

R4.3 Modal analysis

Method: generalised eigenvalue problem $K \cdot \phi = \omega^2 \cdot M \cdot \phi$ on the free DOFs.

- **Lumped mass:** translational mass at nodes (half the sub-element mass to each end + mass from load cases); rotational inertia = 0.
- Small-medium models: dense eigen solver; large models: sparse *shift-invert* around 0 for the lowest modes.
- 1–50 modes; sorted from the longest period. Constraints are included in the reduced DOF space ($M_r = T^T M T$).
- Output: period T , ω , mode shapes, mass participation ratios $UX/UY/UZ$ and their cumulative sums. Mode-shape member forces are not reported. Mode shapes are mass-normalised ($\phi^T M \phi = 1$) and, for display, normalised to a

1 m translational peak.

Accuracy: uniform cantilever, error of mode 1 period $\approx 1.3\%$ with 6 sub-elements (0.46 % with 10); fixed-fixed 0.04 %.

Remember: modal analysis is always linear with full stiffness (or modifiers). P-Delta and the direct analysis method do not affect it. Tension/compression-only members are **all active** in modal analysis — the lateral stiffness of tension-only X bracing is counted twice; note this when periods are used for design.

R4.4 Diaphragm centre of mass and centre of rigidity

Every diaphragm (a constraint group tying u_x , u_y and r_z , all nodes at one elevation) is reported in the **Diaphragms** tab:

- **Centre of mass** (XCM, YCM): translational masses of the group nodes from the Mass Source. Masses at nodes inside members and at offset ends are first folded to both member joints with the lever rule.
- **Centre of rigidity** (XCR, YCR): on that diaphragm only (other floors free) unit F_x , F_y and M_z are applied and the floor rotations θ_x , θ_y , θ_z are read. A force through the centre of rigidity does not rotate the floor: $y_{CR} = y_{Fx} + \theta_x/\theta_z$, $x_{CR} = x_{Fy} - \theta_y/\theta_z$. Full linear stiffness (modifiers included; P-Delta/tension-compression not).
- In the views: ● centre of mass, ⊕ centre of rigidity.

R5. Non-linear analysis

Non-linear analysis **cannot be superposed**. Load cases are still solved linearly (first order), while **each combination is analysed on its own** with its factored loads. Envelopes and Check Design read these non-linear combination results.

R5.1 P-Delta (second order)

Purpose: the effect of axial load on the deformed shape — **P-Δ** (sway between member ends) and **P-δ** (curvature along the member).

Theory: $K_t = K + K_g(N)$. K_g = the **consistent geometric stiffness matrix** per sub-element (McGuire, Gallagher & Ziemian, *Matrix Structural Analysis*, eq. 9.19 without the end-moment and torsion terms), proportional to the axial force N : compression softens, tension stiffens. Because every frame is divided into 6 sub-elements, $P-\delta$ is represented.

Iteration per combination: start from the linear solution $\rightarrow N$ of every sub-element; assemble $K + K_g(N)$, solve u , compute the new N ; repeat until the change in $N \leq 1 \times 10^{-4}$ relative to the largest $|N|$ (max. 50 iterations). Member forces: $p = k \cdot u - f + N \cdot (k_g \cdot u)$; released members are condensed with the same $k + N \cdot k_g$.

Buckling detection: when $K + K_g$ is not positive definite (pivots of the LDL^T factorisation, Sylvester's inertia test), the run stops with "structure unstable".

Output: number of iterations and the **amplification factor** = largest ratio of second-order / first-order sway between column ends ($\approx B2$), shown in the table header.

Stiffness: full elastic (with modifiers). The torsional term $N \cdot I_p / A$ is deliberately left out of K_g : without warping stiffness, that term gives false torsional buckling of open sections.

Meaning for steel design: P-Delta without the direct analysis method = the **effective length method** (SNI 1729 Appendix 7) — design still uses the K factor.

R5.2 Direct analysis method — SNI 1729:2020 Chapter C

Component	Clause	Implementation	Automatic?
Second order P-Δ + P-δ	C2.1	P-Delta engine R5.1 (forced on)	Yes
Reduced stiffness	C2.3(a)	Steel frames $\times 0.8$ on EA, EI and GJ — in combinations only	Yes
$\tau_b = 1 +$ additional notional	C2.3(c)	$0.001 \cdot Y_i$ at every node, in the direction of the resultant horizontal load of the combination	Yes
Notional $0.002 \cdot Y_i$	C2.2b	Notional load cases (R2.5) in gravity combinations $\pm X / \pm Y$	No — the user creates the cases and combinations (the Run dialog warns)

- Steel frame = a steel section, or no shape with a steel material. Concrete is **not** reduced by 0.8. The 0.8 factor multiplies any existing modifier.
- Load cases and modal analysis use full stiffness.
- Steel design: $K = 1$ (per-frame K overwrites still apply); warnings for combinations without notional loads, $\Delta_2 / \Delta_1 > 1.7$ without notional loads (§ C2.2b(4)), and concrete frames without cracked modifiers.
- DAM combination displacements use reduced stiffness and notional loads — **not for serviceability checks**.
- Not provided: variable τ_b , automatic notional loads without load cases.

R5.3 Second order for concrete — SNI 2847:2019 § 6.7

SNI 2847:2019	Content	In the application
§ 6.6.4 Moment magnification	First-order moment $\times \delta_{ns} / \delta_s$	δ_{ns} non-sway in concrete design; sway δ_s not — use P-Delta
§ 6.7 Elastic second-order analysis	P-Δ + P-δ with the reduced stiffness of § 6.6.3.1.1 (beams 0.35I _g , columns 0.70I _g)	P-Delta + cracked property modifiers
§ 6.8 Inelastic second order	Material non-linearity	Not available

- The steel reduction factor 0.8 comes automatically from the DAM option; concrete from **modifiers entered by the user** (the 0.35 / 0.70 buttons only fill in the factors). The concrete design report warns when concrete frames do not use I22 and I33 < 1.
- Applying reduction factors alone is **not** a second-order analysis; second order happens only when P-Delta is ticked.
- **Recommendation:** pure concrete structures → P-Delta + cracked modifiers (without DAM). Mixed steel–concrete structures → DAM is acceptable.
- Not applied: stiffness of $1.4 \times$ cracked for service deflection/drift (§ 6.6.3.2.2) — service drift is conservative; division of I by $(1 + \beta_{ds})$ for sustained lateral loads.

R5.4 Tension-only / compression-only

- Limit **0 kN**; other limits are not available.
- Load cases and modal: linear, all members active. Combinations: non-linear with active/inactive member states.
- Inactive member = **axial release at one end** (bottom for tension-only, top for compression-only); bending remains.
- State from the trial force $EA/L \times$ elongation between joints, dead band $10^{-3} \times$ the largest |N|; iterated until the states stop changing.
- Can be combined with P-Delta / DAM.
- A structure that becomes unstable after members go inactive → error message; alternating states → "not converged".
- Not allowed together with a P release. Not available: gap/hook, cables, compression-only supports.

R5.5 Run options

Static	P-Delta	DAM	Combinations computed as
X	–	–	No static (modal only): no member forces, reactions, envelopes, design
✓	X	X	Linear, superposition of load cases
✓	✓	X	Second order, nominal stiffness (+ modifiers)
✓	✓ (locked)	✓	Second order + 0.8 steel stiffness + 0.001·Y _i notional

Tension/compression-only members make combinations non-linear even when P-Delta is not ticked. ASD steel design with a second-order run: [R6.7](#).

R6. Steel design to SNI 1729:2020

R6.1 Workflow

SNI 1729:2020 = AISC 360-16; clause numbers follow AISC. Method **LRFD** (ϕR_n) or **ASD** (R_n/Ω). **Check** mode (sections already chosen), not automatic section selection.

1. Combinations: a group marked design; otherwise all strength (LRFD) or service (ASD) combinations.
2. Member forces from the run (first order, P-Delta or DAM — **R6.6**). Changing design parameters needs no re-run.
3. **Check points** of every frame: every station, the $\frac{1}{4}$ points between stations, and the moment peaks inside sub-elements.
4. D/C = the largest ratio over all combinations and points; default limit 1.0.

R6.2 Parameters and unbraced lengths

Parameter	Default
ϕ flexure / shear / compression / tension yielding	0.9
ϕ tension rupture	0.75 (tension rupture not checked)
Global K	1
Lx, Ly, Lb	Between brace points from the model (below); without brace points = frame length (with offsets: flexible length)
Cb	From the moment diagram of each unbraced segment (F1-1)

Per-frame overwrites: ratios Lx, Ly, Lb, Kx, Ky, Cb. A ratio < 1 divides the frame into uniform segments of ratio·L from node i and replaces the automatic brace points; Cb per segment. Cantilevers (Cb = 1, K) are set manually.

Brace points from the model: frame ends are always brace points. Interior brace points = nodes on the frame where **other frames connect** or **a support restrains translation**. Their direction ($|\cos| \geq 0.3$) relative to the local axes of the checked frame:

Direction of the frame / support at the node	Braces
Along local axis 2 (e.g. a post on a beam)	Lx (strong-axis buckling)
Along local axis 3 (e.g. a secondary beam into a girder)	Ly (weak-axis & torsional buckling) and Lb (LTB)
Collinear with the frame	Does not brace

Frame ends without any bracing (cantilever tips, or frames continued by a collinear frame) get a note: an unbraced length running on into the next frame must be overwritten. Bracing by floor slabs is not counted.

Fy/Fu from the material (BJ37: 240/370 MPa). A steel material without Fy → frame "not checked".

R6.3 Strength per section type

Section	Compression	Flexure	Shear	Other
WH / I	E3, E7 (slender web)	F2 (compact, LTB with Cb), F3 (noncompact flange), F4/F5 (noncompact/slender web), F6 (weak axis)	G2.1; G2.2 tension field with stiffeners	J10 optional
UNP	E4 (flexural-torsional)	F2 channel: $c = (h_o/2)\sqrt{I_y/C_w}$; F6 with $b = b_f$	G2.1(b), $\phi = 0.9$	eo and Cw from thin-wall formulas

Section	Compression	Flexure	Shear	Other
SHS/RHS	E3, E7	F7 compact / noncompact / slender; LTB F7.4 only RHS strong axis; slender web F7.3(c)	G4	Torsion H3.1; H3-6 when $T_r > 0.2T_c$
Pipe	E3, E7-2	F8, resultant moment	G5 with L_v from the shear diagram	Torsion H3.1(a)
Angle	E5(a)/(b) + E7, long or short leg	F10 principal axes (yielding, LTB F10-4, leg local buckling), β_w from polygon integration, $C_b = 1$	G3	Interaction H2
CT	E3, E4-3 (shear centre at mid-thickness of the flange), E7	F9 by the sign of M_3 (stem in tension/compression), y axis like F6	G3, $k_v = 1.2$	Interaction H1

Interaction: H1-1a / H1-1b for symmetric sections; H2 for angles; H3-6 for torsion of closed sections. **Tension:** yielding of the gross section only.

R6.4 Assumptions per section type

- **UNP:** load through the shear centre / member restrained against twist (F2 assumption). e_o is shown in the report; torsion from eccentricity is not computed.
- **Angles:** compression E5(a) planar truss, E5(b) space/box truss; connected through the long leg (default) or the short leg ($E5(a)(2) + 4[(b/b_s)^2 - 1] \geq 0.95L/r_z$; $E5(b)(2) + 6[(b/b_s)^2 - 1] \geq 0.82L/r_z$). E5 conditions: $L_c/r \leq 200$, $b/b_s < 1.7$, $b/t \leq 0.71V(E/F_y)$; when violated and the angle is in compression, the frame is "not checked". Geometric-axis moments are projected conservatively to the principal axes: $|M_3 \cdot \cos \theta| + |M_2 \cdot \sin \theta|$.
- **HSS/pipe:** thickness = nominal JIS thickness (no 0.93t reduction).
- **Open sections (WH/UNP/angle/CT):** torsion H3.3 — St Venant shear stress only ($\tau = T \cdot t/J$ added to the shear ratio); warping normal stresses are not computed.
- **Stiffened girders:** transverse stiffener spacing per frame $\rightarrow$ F13.2, k_v G2.1, tension field G2.2 in interior panels. When the stiffener plate size is entered: $G2.3 (b/t)_{st} \leq 0.56V(E/F_y)$ and $I_{st} \geq I_{st2} + (I_{st1} - I_{st2})p_w$. Stiffener welds are not checked.
- **Concentrated forces J10** (I sections, enabled per frame): end reactions or also internal forces (V2 jumps). Without bearing stiffeners: web local yielding J10.2 and web crippling J10.3. With a pair of bearing stiffeners: plate column + web strip 25tw (12tw at the end), $L_c = 0.75h$, bearing J7-1, J10.8 requirements. J10.1, J10.4–J10.7 not checked (noted in Detail).
- **CNP, lipped channel LC, lipped Z** (cold-formed, SNI 7971): not designed.

R6.5 Not checked

Noncompact channels (F2); I webs beyond the F13.2 limit; tension rupture (A_e); stiffener welds; J10.1, J10.4–J10.6; warping normal stresses; angles in compression outside the E5 conditions (Chapter H); B1/B2 amplification of Appendix 8 (second-order effects only through P-Delta/DAM).

R6.6 Second-order effects and design

Run	Member forces	K factor	Note in the report
Linear	First order	Global K / overwrite	Second-order amplification not yet included
P-Delta	Second order, full stiffness	Global K / overwrite	Effective length method (Appendix 7)
DAM	Second order, 0.8 stiffness + notional	K = 1 (K overwrite kept)	Warnings on notional loads, $\Delta_2/\Delta_1 > 1.7$, concrete without modifiers

R6.7 ASD

Available strength R_n/Ω (B3.2) with the fixed Ω of SNI 1729:

Limit state	Ω
Tension yielding D2, compression E, flexure F, shear G (except below), torsion H3, stiffener column J4.4	1.67
Shear G2.1(a) rolled I sections, web local yielding J10.2	1.50
Web crippling J10.3, bearing J7	2.00

Combinations = *service* combinations or a design group; interaction as in LRFD. **Second order with ASD** (C2.1(4)): service combinations are analysed at $1.6 \times$ their loads and the results divided by 1.6, so the second-order amplification is at the factored level; as a consequence, service-combination deflections in such a second-order run are slightly larger than the true service deflections.

R7. Concrete design to SNI 2847:2019

R7.1 Workflow and parameters

SNI 2847:2019 = ACI 318-14 (V_c without the size factor; ϕ from the tensile strain ϵ_t between ϵ_y and 0.005). The roof-joint exception is taken from ACI 318-19 § 18.7.3.2.

- **Design** mode (required reinforcement) for beams and columns; columns can also be in **check** mode with provided bars.
- Sections: rectangle $b \times h$ and circle D . Frames inclined $> 45^\circ$ are designed as **columns**, otherwise as **beams**; can be overwritten. Circles are always columns.
- Combinations and check points as for steel. **Design uses the gross section.**

Parameter	Default
Frame system	SMF / SRPMK (options OMF / IMF / SMF)
ϕ flexure / shear / torsion	0.9 / 0.75 / 0.75
ϕ compression tied / spiral	0.65 / 0.75
ϕ joint shear	0.85
f_y / f_{yt}	420 / 280 MPa
Cover beams / columns	40 mm
Diameter stirrups / beam bars / column bars	10 / 19 / 19 mm
Beam design at the column face	On

R7.2 Beams

- **Flexure M3**: singly reinforced, doubly reinforced when needed ($\epsilon_t \geq 0.005$). $A_{s,min}$ § 9.6.1.2 or $4/3 A_s$ required (§ 9.6.1.3).
- **Shear V2**: $V_c = 0.17\sqrt{f'_c} \cdot b_w \cdot d$ with the effect of axial force (§ 22.5.6 / 22.5.7), A_v/s , minimum stirrups § 9.6.3.3.
- **Torsion**: threshold T_{th} , A_t/s , $A_l \geq A_{l,min}$ ($\theta = 45^\circ$), section limit § 22.7.7.1; no compatibility-torsion reduction. The *Ignore beam torsion* option designs beams for flexure + shear only.
- Axial force is ignored for flexure (reported when $> 0.1f'_c A_g$); included in V_c and the torsion limit.
- **Envelope**: each combination is evaluated at the moment peaks of the other combinations.
- **Support face**: design starts half the column dimension from the node (when on). Offset ends are already the face.
- Proportioning rules: IMF § 18.4.2.2, SMF § 18.6.3.2 (bottom A_s at the ends $\geq \frac{1}{2}$ top A_s ; anywhere $\geq \frac{1}{4}$ max end A_s), SMF § 18.6.3.1 (top & bottom $\geq \min(A_{s,min}, 4/3 A_s)$); max ρ 4 % (SMF 2.5 %).
- No effective slab width (T beams).

R7.3 Columns

- **Biaxial P-M2-M3 interaction** from strain compatibility with an inclined neutral axis: one-quadrant surface of 17 angles $\times$ 27 depths c + tension/compression poles. D/C = ratio along the line from the origin to (P_u , M_{u2} , M_{u3}) relative to the surface.
- Upper limit $\phi \cdot 0.80 \cdot P_0$ (tied) / $\phi \cdot 0.85 \cdot P_0$ (spiral); pure tension $\phi \cdot f_y \cdot A_{st}$.
- **Required ρ** found by bisection 1–8 % (SMF 6 %); bars assumed evenly around the perimeter, centre spacing ≤ 150 mm (circles: $\lceil 2\pi R/150 \rceil$, minimum 6).
- **Shear V2/V3**: A_v/s for SMF. Circles: $b_w = D$, $d = 0.8D$; spiral p_s min § 25.7.3.3 / SMF § 18.7.5.4.
- **Check mode** (provided bars in the section: diameter, number per side or around the perimeter, spiral): P-M2-M3 D/C with those bars; shear and joints use those bars; warning when ρ is outside the limits.

- **Slenderness:** non-sway columns (default) magnified by δ_{ns} (§ 6.6.4.5; limit § 6.2.5, k default 1, $(EI)_{eff} = 0.4Eclg/(1 + \beta_{dns})$, C_m , $M_{2,min}$, $\delta \leq 1.4$). Sway columns with $k \cdot l_u/r > 22$ need P-Delta (δ_s not provided).
- Column torsion is not designed.

R7.4 Provided bars and drawings

From the required reinforcement bars are chosen and written as **4D19 / Ø10-150** ($\emptyset$ = plain $f_y \leq 280$ MPa, D = deformed).

- **Beams** — three zones: support = $\max(l_n/4, 2h \text{ for IMF/SMF})$ from the face ($\leq l_n/2$), span in between. Top/bottom bars = the largest required area + torsion A_l $\frac{1}{4}$ top, $\frac{1}{4}$ bottom ($\frac{1}{4}$ per side $\rightarrow$ side bars), minimum 2; layers with clear spacing $\max(25, db)$ — a second layer gets a warning. Stirrups $s = A_b / \max(A_v/s \text{ per leg} + A_t/s)$, rounded down to 25 mm, limited to $d/2$ ($d/4$ when $V_s > 0.33\sqrt{f_c'} \cdot b_w \cdot d$), $\phi_h/8 \leq 300$ (torsion), support zone SMF $\min(d/4, 6db, 150)$ / IMF $\min(d/4, 8db, 24ds, 300)$; $s < 50 \rightarrow "$ Ø10-?" + warning.
- **Columns** — the number of bars covers the required A_s , then bars are added on the side that most reduces D/C until $D/C \leq 1$; D/C is recomputed with the provided bars. Ties at every corner and alternate bar (§ 25.7.2.3; SMF $h_x \leq 350$). Spacing within ℓ_o : SMF $\min(b/4, 6db, so)$, IMF $\min(8db, 24dt, b/2, 300)$; outside ℓ_o : SMF $\min(6db, 150)$, others $\min(16db, 48dt, b)$.
- **Drawings** (Detail): beams — support and span sections, stirrup elevation; columns — section and **P-M interaction diagram** (design surface ϕ and nominal at the governing angle / M_3 / M_2 , points = all combinations).

R7.5 Capacity design (IMF / SMF)

Check	SMF	IMF
Beam shear	$V_e = (M_{pr,i} + M_{pr,j})/l_n + V_g$; M_{pr} with $1.25f_y$; V_g = simple-span gravity shear. $V_c = 0$ within $2h$ when $V_p \geq \frac{1}{2}V_e$ and $P_u < A_g \cdot f_c'/20$	$\min(\text{shear from } M_n, \text{shear with earthquake} \times 2), \geq \text{analysis } V_u$
Column shear	$V_e = 2M_{pr}/l_u$, limited by ΣM_{pr} of the beams at the joint (§ 18.7.6.1.1)	As for beams
Strong column–weak beam	$\Sigma M_{nc} \geq 1.2 \cdot \Sigma M_{nb}$ per joint and plane (§ 18.7.3.2); roof joints with $P_u < A_g \cdot f_c'/10$ excluded	—
Joint shear	$V_j = 1.25f_y(A_{s1} + A_{s2}) - V_{col}$; $\phi V_n = 0.85 \cdot \gamma \cdot \sqrt{f_c'} \cdot A_j$	—

V_e is computed only for combinations with earthquake (all combinations when the model has no earthquake combination). M_{pr} and M_n come from the **required reinforcement** (singly reinforced, without slab).

R7.6 Second-order effects

Without P-Delta: first-order forces with δ_{ns} for slender non-sway columns (**R7.3**). With P-Delta: elastic second-order analysis § 6.7 — cracked stiffness entered through modifiers; the report states whether all concrete frames use cracked sections (**R5.3**).

R8. Summary of limitations, validation and verification

R8.1 Main assumptions and limitations

Topic	Limitation	Reference
Elements	Euler-Bernoulli frames without shear deformation and warping; slabs only distribute loads (no meshed plate elements)	R1.3 , R2.8
Supports	No inclined or compression-only supports	R1.9
Loads	No uniform moments about axes 2/3, temperature loads $\alpha \Delta T$, live-load patterns	R2.1 , R2.4
Earthquake	No accidental torsion, storey drift check or θ	R3.5
Modal	Full linear stiffness; tension-only braces taken as active	R4.3
Non-linear	Geometry (P-Delta) and tension/compression-only with limit 0 only; no material non-linearity	R5
Concrete second order	One set of modifiers for all analyses	R5.3
Steel	Tension rupture, B1/B2, warping, part of J10, cold-formed steel not checked	R6.5
Concrete	No T beams, sway δ_s , column torsion, development/splice lengths	R7

R8.2 Validation

Every solver and design feature is tested against closed-form solutions and hand calculations, and the main features were re-checked by independent reviewers who wrote their own reference solvers without seeing the application code (P-Delta, direct analysis method, tension-only, end offsets, ELF/RS earthquake, load-distributing areas, steel and concrete design). Theory references: Timoshenko & Gere, *Theory of Elastic Stability*; McGuire, Gallagher & Ziemian; AISC 360-16 Commentary; AISC Steel Construction Manual tables.

R8.3 Verification against commercial analysis programs

Analysis and design results were compared with a commercial structural analysis program on 19 test models (beams and frames with every load type, releases, local-axis rotation, diaphragms, modal, P-Delta, direct analysis method, tension-only, end offsets, partial releases, steel and concrete design, secondary beams at the middle of girders, load-distributing areas, support springs) and through two-way .s2k file exchange. Analysis results are identical within numerical precision; design differences are explained down to the formulas. Details, comparison tables and the list of features not yet compared are in the report *Analysis Verification EDFS Matrix*; a summary is in the application under **Help → Legal → Verifikasi**.