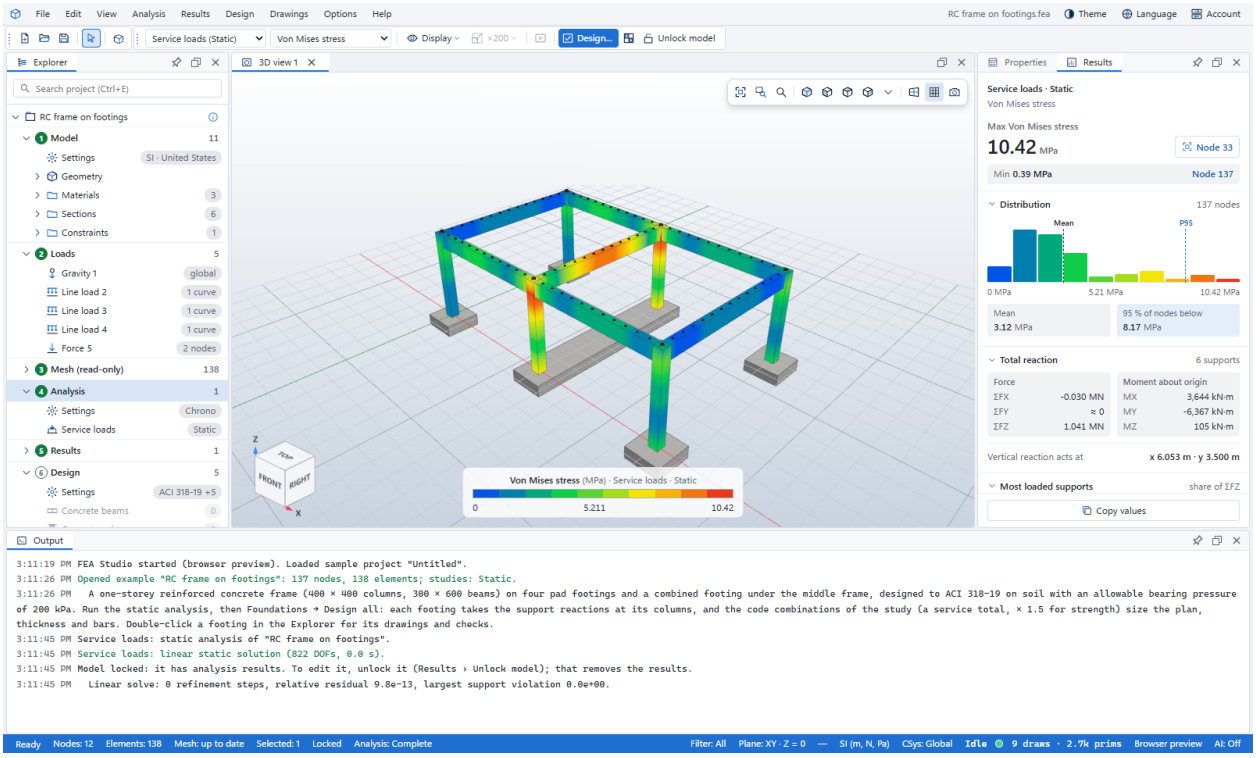
FEA STUDIO

User Manual

Part 2 of 5 · Analysis

Set up studies (linear static, modal, linear dynamic, nonlinear static and response spectrum), run them, and read displacements, stresses, member forces and reactions.



The manual set

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

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

Contents

About this part ... 3
Studies ... 4
Analysis types ... 4
Adding a study ... 4
Study settings ... 5
Analysis settings ... 5
Response spectrum studies ... 5
Load cases and combinations ... 7
The mesh ... 8
Solvers ... 9
High-order solids (MFEM) ... 9
Running an analysis ... 10
Ways to run ... 10
What happens when you run ... 10
The status bar ... 11
Locked models ... 11
Reading the results ... 12
The Results toolbar ... 12
Quantities ... 13
Deformed shape ... 13
Animation ... 14
The legend ... 14
Colour scales ... 14
The Results panel ... 15
Reactions ... 15
Results table ... 15
Charts ... 15
Several result sets ... 16
Exporting results ... 17
Troubleshooting ... 18

About this part

This part starts from a finished model, as described in Part 1 (Modeling). It covers:

- how to set up **studies**, which are the analyses you want to run;
- the five analysis types and their settings;
- load cases and load combinations in the results;
- the mesh and the solvers, including high-order solid elements;
- how to **run** a study;
- how to **read the results** in the 3D view, the Results panel, tables and charts;
- how to export results.

Part 3 (Design) continues from a solved static study. In this part, menu commands are written as **Menu › Group › Command**, and keys are shown like F5.

Studies

A **study** is one analysis of the model: its type and its settings. A project can hold several studies, for example a linear static study for design and a modal study for the natural frequencies. Studies are listed under stage **4 Analysis** in the Explorer. Each one shows its type as a badge, and the stage shows whether it has been run.

Analysis types

FEA Studio has five analysis types. All of them run on the built-in **Project Chrono** solver; linear static and modal studies of solid models can also run on **MFEM** with high-order elements (see "Solvers").

Analysis menu	Study type	What it does
Linear static	Static	Linear elastic analysis with small displacements: one solution per load case, or one combination
Modal	Modal	Natural frequencies, periods and mode shapes
Linear dynamic	Dynamic	Linear transient response to loads applied at t = 0, integrated in time (HHT)
Nonlinear static	Nonlinear	Large-displacement static analysis in load increments, with Newton–Raphson iterations in each
Response spectrum	Response spectrum	Modal combination (SRSS or CQC) of a code design spectrum

Note FEA Studio has no separate buckling or P-Δ study type. Use a Nonlinear static study when large displacements matter; the steel design settings offer member and storey sway amplification (Part 3).

Adding a study

1. In the Explorer, right-click the **Analysis** stage and choose **Add study**.
2. In **Add study**, type a **Name**. It follows the type until you edit it, for example "Static 2".
3. Choose the **Analysis type**.
4. For Static and Modal, choose the **Solver**: **Project Chrono** (beams, shells and solids) or **MFEM** (higher-order solid elements, for solid models only).
5. Click **Add**. The new study is selected so you can set its options in Properties.

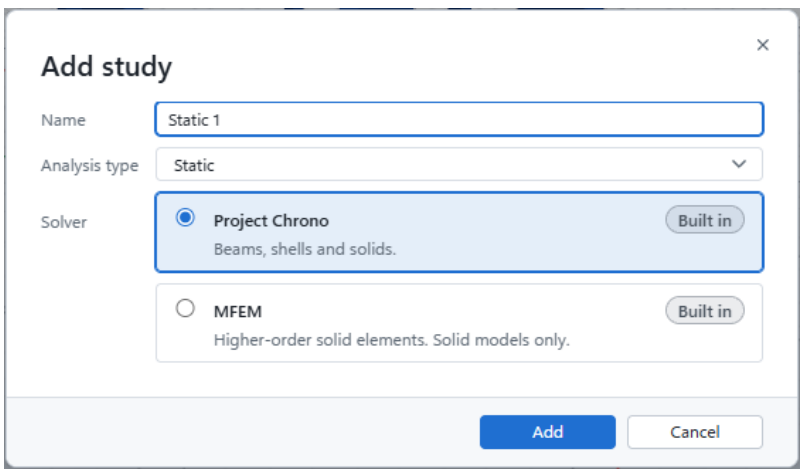


Figure 1. The Add study dialog: name, analysis type and, for static and modal studies, the solver

You can also skip this dialog. **Analysis › Analysis › Linear static** (or Modal, Linear dynamic, Nonlinear static, Response spectrum) runs the first study of that type, and creates one with default settings if none exists.

Right-click a study for **Run study**, **Rename** (F2), **Delete** and **Properties**. Drag studies to change their order.

Study settings

Select a study to see its settings in **Properties**. The **Study** group holds the name and the analysis type; the type can be changed later. The other settings depend on the type:

Setting	Study types	Meaning (default)
Solve	Static	**Each load case**: one solution per load case, plus the code load combinations and their envelopes; the design modules use these. **One combination**: a single solution of the combination below.
Load combination	Static (one combination), Dynamic, Nonlinear, Response spectrum	For example `1.35 Dead + 1.5 Live`. Empty means every load once.
Mass from load cases	Modal, Response spectrum, Dynamic	For example `1 Dead + 0.25 Live`. Self-weight is always counted as mass.
Number of modes	Modal	Lowest modes to find (6)
Duration · Time step	Dynamic	1 s · duration / 200. Use a step of about 1/20 of the shortest period of interest.
Rayleigh β	Dynamic	Stiffness-proportional damping in seconds; the damping ratio is about β·ω/2 (0 = undamped)
Member force history	Dynamic	Keep member forces at every output frame (larger result files)
Plastic hinges	Dynamic	Plastic hinges at the ends of tubular members (plastic moment with the N–M interaction), for blast and impact; with a limit displacement
Load increments	Nonlinear	Load steps, each solved with Newton–Raphson (10)
Solver · Element order	Static, Modal	Project Chrono or MFEM; with MFEM, the element order: 1 = linear, 2 = quadratic (default), 3–4 = cubic, quartic

> **Tip** For design, use a Static study with **Solve: Each load case**. Each load case is then solved once, and the code combinations and envelopes are built from those solutions without solving again. That is what the design modules need.

Analysis settings

Expand the **Analysis** stage in the Explorer and double-click **Settings** (or choose **Analysis settings…** from it). The **Analysis settings** dialog lists every study with its **Solver**. A study whose type only one solver can run says so. Changing a solver does not remove results already run. The badge beside **Settings** shows the solvers in use, for example **Chrono**.

Response spectrum studies

A response spectrum study is made from the code spectrum:

1. Choose **Assign › Generate › Response spectrum**.
2. Pick the code, then enter the seismic inputs (as for **Earthquake loads**; see Part 1).
3. In **Response spectrum study**, set the options below, then click **Create**.

Field	Default
Direction factors X Y Z	`1 0.3 0`
Combination of directions	100-30 or SRSS
Modes	30
Damping ratio (CQC)	0.05
Scale up to NN % of the static base shear	On (ASCE 7 and KDS)
Use each direction as an earthquake load case in design	Gives the load cases **Seismic RSA X** and **Seismic RSA Y**

The study is named **Response spectrum**. Running the command again updates it. **Analysis › Analysis › Response spectrum** runs the first response spectrum study, or makes one from the code spectrum first.

Load cases and combinations

Every load of the model belongs to a **load case**, and every case has a category such as Dead, Live, Wind or Earthquake (Part 1). The analysis uses them in two ways:

- A Static study solved **Each load case** gives one result per load case, and builds the load combinations of the design code from those results.
- A study with a **Load combination** (for example `1.2 Dead + 1.6 Live`) solves that one combination. A study with no combination applies every load once.

Define › Load cases › Load combinations chooses which code combinations are used and lets you add your own (see Part 3). The dialog stays available after solving: when you change it, the combination results are rebuilt at once, without solving again.

In the results, the case selector is grouped as follows:

- **Load cases**
- **Strength combinations**
- **Service combinations**
- **Envelopes**

Your own combinations are marked **(yours)**. An envelope shows, for each value, the largest or smallest result over the strength or service combinations. The deformed shape of an envelope is therefore not one real load state.

The mesh

The solvers work on the mesh, not on the geometry you draw. You do not have to build it yourself: when you run a study, the mesh is generated first if there is none, or if the model changed since it was built (Part 1 describes the mesh settings).

- Beams, trusses and cables become line elements; surfaces with a thickness become shell elements; solids become tetrahedra or hexahedra.
- **Mesh › Generate › Settings** sets the element size, quadrilateral or triangular shells, and hexahedral or tetrahedral solids.
- **Mesh › Inspect › Check quality** opens the **Mesh quality** panel. Check the worst elements of a solid model before you trust its stresses.
- The status bar shows **Mesh: none**, **out of date** or **up to date**.

Solvers

Solver	Kind	Used for
Project Chrono	Built in	Beams, trusses, cables, shells and solids: Linear static, Modal, Linear dynamic, Nonlinear static and Response spectrum
MFEM	Built in	Higher-order solid elements (order 1–4) for Linear static and Modal. Solid models only: tetrahedra and hexahedra, no beams or shells.
CBFEM	Built in	Steel connections: a shell model of the plates, bolts and welds of each connection (Part 3)

High-order solids (MFEM)

Linear elements are stiff in bending and under-estimate stress peaks unless the mesh is very fine. MFEM solves the same tetrahedral or hexahedral mesh with higher-order elements, which gives far more accurate stresses in bending and at stress concentrations.

1. Mesh the solids (Part 1). Every element must be a tetrahedron or a hexahedron.
2. Add a Static or Modal study and choose **MFEM** as the **Solver**, or change the solver of an existing study in its Properties or in **Analysis settings**.
3. Set the **Element order** in the study's Properties: 2 (quadratic) is the default. Each step up multiplies the number of equations by about 2–3 for tetrahedra and 3–4 for hexahedra.
4. Run the study as usual.

If the model also has beams, trusses or shells, the Solver field says that MFEM handles solid models only; use Project Chrono for those models.

Note In the browser version, analyses are sent to the local solver server. Without it, a run stops with the message that it needs the desktop app or the solver server.

Running an analysis

Ways to run

Command	Runs
F5 · Analysis › Run › **Run analysis**	The study selected in the Explorer, or else the first static study
Run linear static ▾ on the toolbar	The first linear static study; the drop-down picks another type
Analysis › Analysis › **Linear static / Modal / Linear dynamic / Nonlinear static / Response spectrum**	The first study of that type, made with defaults if none exists
Right-click a study › **Run study**	That study
Run analysis in the empty Results panel	As F5
Analysis › Run › **Analyse and design**	Makes the design objects, solves the load cases and designs everything in one go (meant for models imported from IFC; see Part 3)

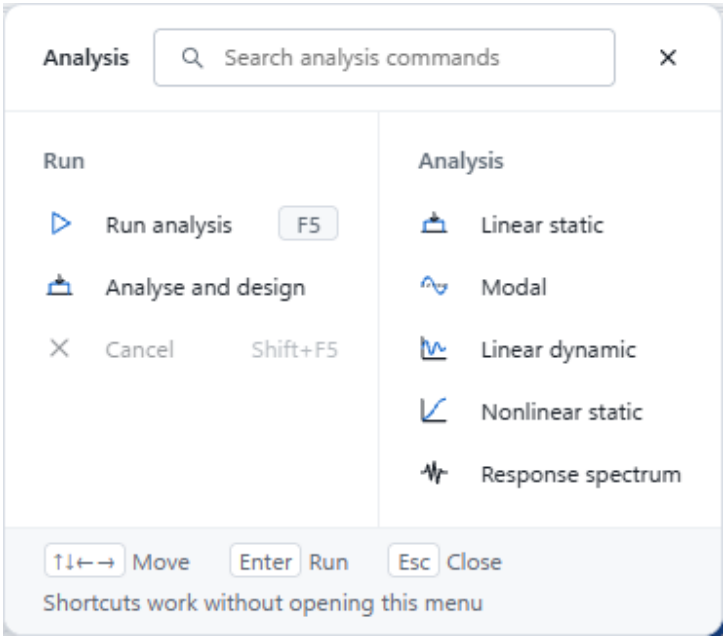


Figure 2. The Analysis menu

What happens when you run

1. **The mesh is checked.** If there is no mesh, or the model changed since it was built, the mesh is generated first. The Output panel says so.
2. **The model is validated.** Problems such as missing sections or no supports are listed in the Output panel. If there is an error, a message names the study and the problem, and nothing runs.
3. **A progress card appears.** It shows the percentage, the current stage, the elapsed time and an estimate of the time left. Click **Cancel**, or press `Esc` or `Shift+F5`, to stop the run. You can hide the card with ×; the status bar keeps showing the progress, and clicking it brings the card back.
4. **When the run finishes:**
   - the Output panel lists the solver's messages;
   - the **Results** panel opens and shows the new result set;
   - the model is **locked**.

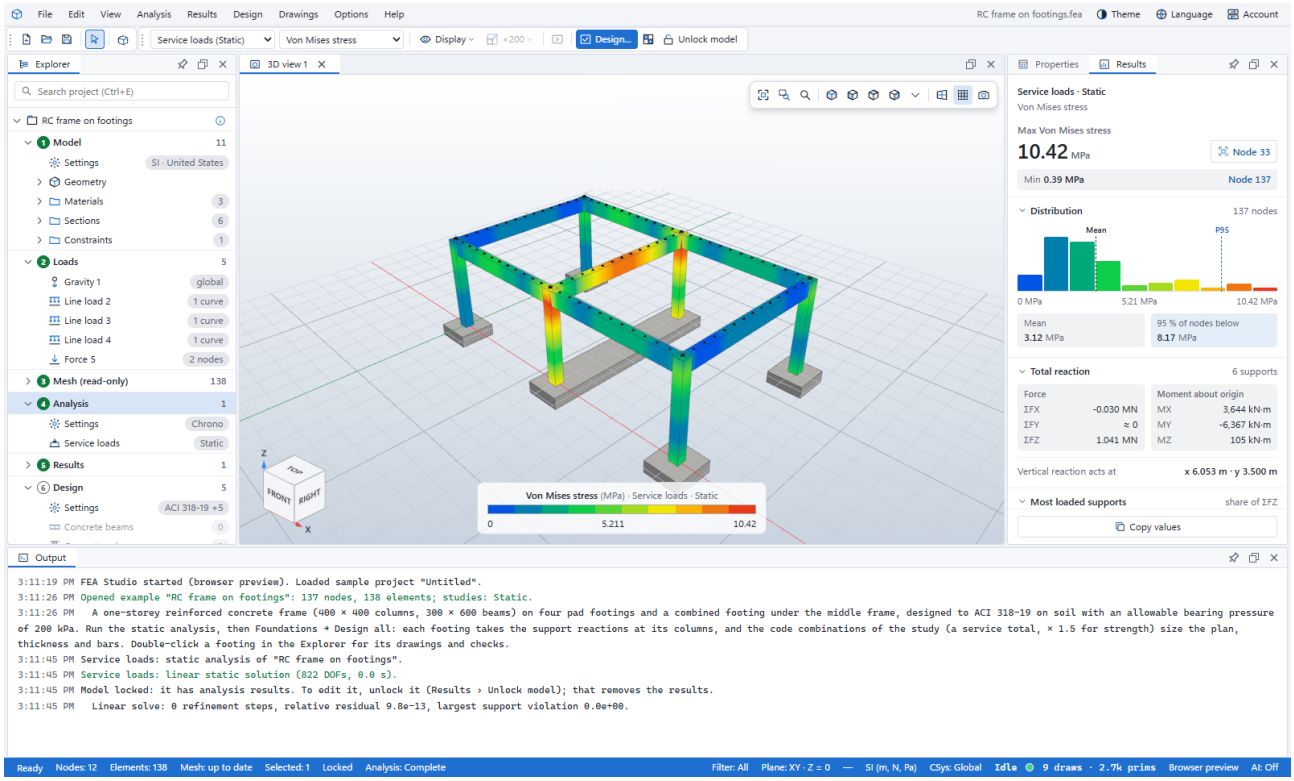


Figure 3. A finished linear static analysis: von Mises stress contours, the Results panel and the solver log

If the run fails or you cancel it, the message is shown in the Output panel and the status bar reads **Failed** or **Cancelled**. Only one analysis can run at a time.

The status bar

Analysis: in the status bar shows **Idle**, **Running…**, **Complete**, **Cancelled** or **Failed**. While a study runs, it shows the study name, the percentage and the stage. Click it to see the progress card, or the Output panel when nothing is running.

Locked models

A model with results is locked, so the results always belong to the geometry you see. The status bar says **Locked**, and editing commands are greyed out.

- **Unlock model** (Results › Lock, or the Results toolbar) asks for confirmation, then removes all result sets and makes the model editable. Solve again after editing.
- Design objects such as beam lines and footings stay, and their settings stay editable while the model is locked.

Reading the results

The Results toolbar

After an analysis, the **Results** toolbar replaces the modelling toolbars. From left to right it has:

Control	What it does
Result set	Chooses which solved study and case to show, or **(none: show the model)**
Quantity	The contour to plot (see the table below)
Case / mode / time	The load case or combination, the mode, or a time slider, depending on the study
Display ▾	Deformed shape, overlay of the undeformed model, reactions, smooth and banded contours
Deformation scale ▾	×N; **Auto** draws the largest displacement at about 8 % of the model size
Animate	Animates mode shapes, dynamic results or the deformation
Design…, D/C ratios	Design (see Part 3)
Unlock model	Makes the model editable again; this removes the results

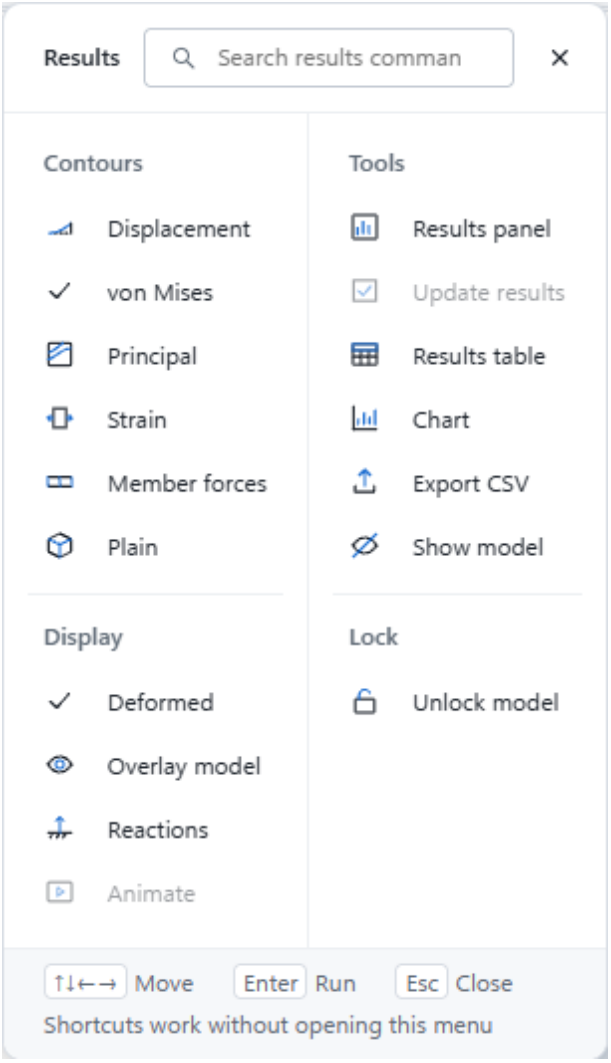


Figure 4. The Results menu: contours, display options, tools and the lock

Quantities

Group	Quantities
Displacement	Displacement |u|, Displacement X / Y / Z, Rotation |θ|
Stress	Von Mises stress, Max principal stress, Min principal stress
Strain	Equivalent strain, Plastic strain
Member forces	Axial force N, Bending moment |M|, Shear force |V|, Torsion T
Motion	Velocity |v|, Absolute acceleration |a| (dynamic studies)
Temperature	Temperature change

Only quantities that the result set contains are listed. The **Results › Contours** menu has shortcuts: **Displacement**, **von Mises**, **Principal**, **Strain**, **Member forces** and **Plain** (the shape without contours).

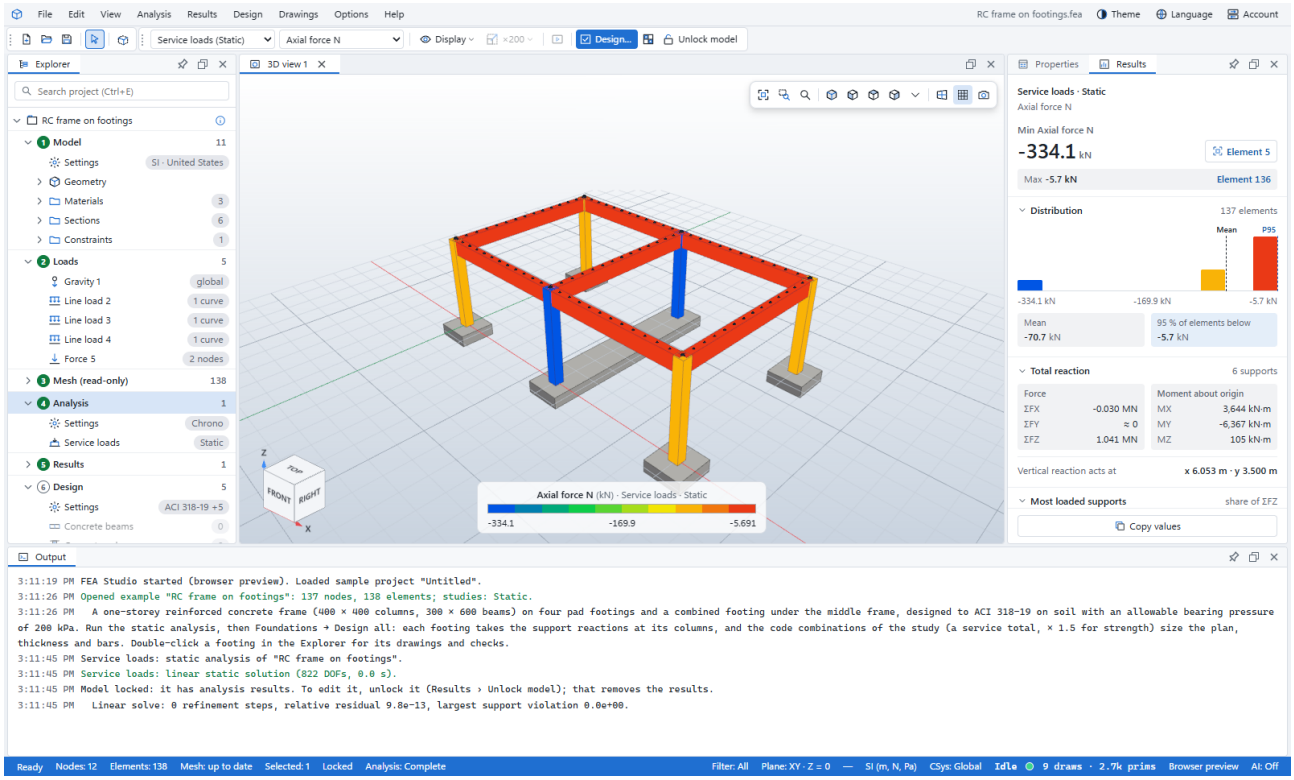


Figure 5. Member forces: axial force N in the beams and columns, drawn on the undeformed model

> **Note** Member forces are shown as colours along the members, not as moment or shear diagrams. For numbers at member ends, use **Results table**, **Export CSV** or the Analysis report.

Deformed shape

FEA Studio draws the model **deformed only when the contour is a displacement quantity, or when there is no contour**. Stresses, strains and member forces are drawn on the undeformed model, so the colours sit where the material actually is. This rule is the reason the **Deformation scale** control is greyed out for stress plots.

- **Display ▾ › Deformed shape** turns the deformation on or off.
- **Overlay undeformed model** shows the original shape as well.

- **Deformation scale** ▾ offers **Auto**, ×1 (true scale), ×10, ×100 up to ×5000, or a custom factor.

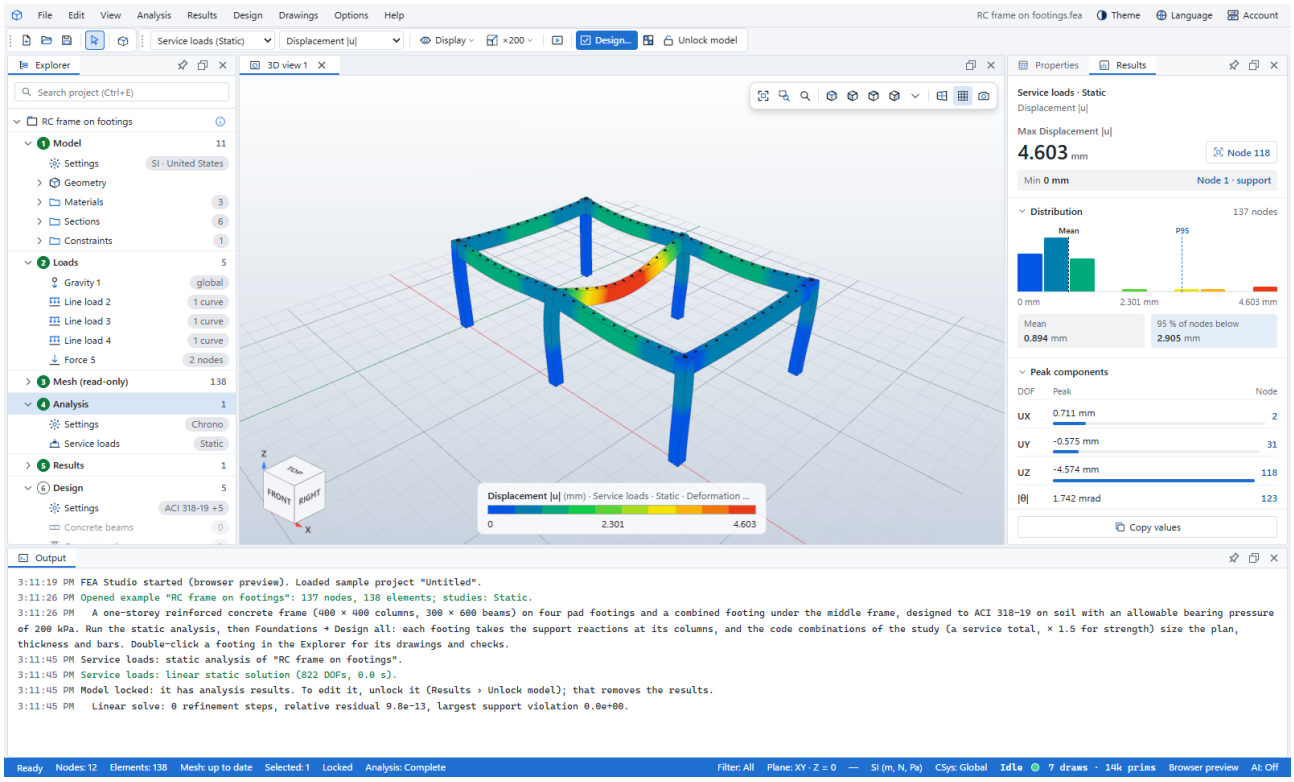


Figure 6. Displacement magnitude with the deformed shape (scaled), and its peak components in the Results panel

Animation

Animate (Results › Display) plays:

- mode shapes swinging back and forth;
- static shapes growing from zero to full;
- linear dynamic and nonlinear static results frame by frame, in a loop.

Animation stops when you switch to a quantity that is not a displacement. Load arrows are hidden while an animation plays.

The legend

The colour legend sits at the bottom centre of the 3D view. It shows:

- the quantity and its unit;
- the study, the case or frame, and the deformation scale;
- the minimum and maximum values.

Hover over the bar to read a value. With **Banded** on, it has 10 bands; otherwise the colours are smooth.

Colour scales

Options › Display › Colour scale chooses the colours of every contour. Your choice is kept for next time.

- **Standard**: FEA Studio (blue–green–yellow–red, the default), Abaqus (12 bands), Rainbow, Jet, Turbo, Cool to warm, Black body, Grayscale.
- **Pastel**: pastel versions of the standard scale, rainbow and cool to warm.
- **Colour-blind safe**: Viridis, Plasma, Inferno, Cividis, Blue–white–red, Blue–orange.
- **Reversed** flips any scale.

Options › Display › Colour map… gives finer control:

- the number of **Bands** (0 = smooth);
- **Logarithmic** scaling;
- the **Range**: Frame shown, All frames, Visible (leaves out hidden and clipped parts), or Custom with your own minimum and maximum.

The Results panel

The **Results** panel (right, beside Properties) summarises the result shown:

- **The headline**: the study and the quantity, the largest value with its unit, and the node or element where it occurs. Click the node or element button to select and zoom to it. The opposite extreme is shown below it.
- **Distribution**: a histogram of the values over the nodes or elements, with the mean and the value 95 % of them stay below.
- **Peak components** (displacements): the largest UX, UY, UZ and rotation, each with its node.
- **Total reaction**: the sums ΣFX, ΣFY, ΣFZ at the supports and their moments about the origin, where the vertical reaction acts, and the most loaded supports. The sums are a quick equilibrium check against the applied loads.
- **Copy values** copies the panel's numbers to the clipboard.

Select a node or element in the 3D view to see its own values in the panel. Sections can be folded away; values too small to matter show as ≈ 0.

Reactions

Results › Display › Reactions draws reaction arrows at the supports. The totals are in the Results panel, and the reactions at every support node are in the Results table, the Analysis report and the CSV export.

Results table

Results › Tools › Results table lists every result of the frame shown, at the nodes or elements. You can sort and filter the columns; the table follows the selection in the 3D view, and it can be saved as CSV.

Charts

Results › Tools › Chart opens a chart in the main area, as a tab beside the 3D view.

Chart	Shows
Frequencies	Natural frequencies of a modal study as bars

Chart	Shows
History / Per mode	A value of the selected node (or the node with the largest value) over time, load increments or modes
Along selection	Values along a path. Select two or more nodes in order with `Ctrl+click`.

The chart can be saved with **CSV** or **SVG**, or sent to **Print**, **PDF** or **Word**.

Several result sets

Each solved study keeps its own result set under stage **5 Results** in the Explorer. Activate a row to show it, or use the **Result set** selector. Right-click › **Remove** deletes a result set; removing the last one unlocks the model. **Results › Tools › Show model** hides the results and shows the plain model, which stays locked.

Exporting results

Command	Output
Results › Tools › **Export CSV**	The frame shown, as a CSV file: node coordinates and displacements; element stresses and strains; beam end forces N, Vy, Vz, T, My, Mz at both ends; support reactions
Results › Tools › **Results table** › CSV	The table as shown, with your sorting and filters
File › Export › **VTK (.vtk)**	The mesh with the shown results, for ParaView and similar tools
File › Export › **Nodes (CSV)**, **Elements (CSV)**	The mesh tables
File › Export › **IFC with results (.ifc)**	The building model with the analysis results attached (Part 1)
File › Export › **Viewport image (PNG)**	A picture of the 3D view as shown
Chart › **CSV**, **SVG**, **Print**, **PDF**, **Word**	The chart and its data
File › Reports › **Analysis report**	The full analysis report (see Part 5)

Troubleshooting

Message or symptom	What to do
"(study name) can't run" with an error in Output	Read the Output panel. Usually a member has no section, a shell has no thickness, or the model has no supports.
Very large displacements, or a "singular" warning	The model is a mechanism: check supports, released ends and unconnected nodes. **Mesh › Display › Show mesh** and **Mesh nodes** help to find gaps.
"An analysis is already running"	Wait for it, or press Esc to cancel it.
Results look stale	The Results panel warns when the mesh differs from the one that was solved. Unlock, then run again.
Can't edit the model	It is locked by its results. Use **Unlock model**.
MFEM is not offered, or says solid models only	The model has beams, trusses or shells, or the study is not Static or Modal. Use Project Chrono.
Browser: "needs the desktop app, or the solver server"	Start the solver server, or use the desktop app.
Netgen not available	Netgen needs the desktop app or the solver server.