Exact Closed-Form Solutions for Multilayer Plates app

Closed-Form Solutions for Multilayer Plates and Shells

User Manual

Document type:	Technical user guide
Interface:	Web-based plate solver
Date:	June 2026

1 Purpose and Scope

MUL^2 Web Solver provides a browser interface for exact closed-form static analysis of simply supported multilayer plates. The interface is intended for fast parametric use: the user defines geometry, loading, laminate data, and an evaluation point, then runs the solver and inspects displacement, stress, tabular, and shear-correction outputs.

The workflow keeps the structural theory selection fixed in the background. Results are reported for CLPT, FSDT, and CUF, with CUF used as the refined layerwise reference in the interface. The advanced theory controls remain part of the development framework but are not exposed in this manual.

This guide describes the graphical workflow. Material stiffness values and load magnitudes must be entered in physical units, as described in Section 6.

2 Access Screen

Before the solver is shown, the access form records the user email, institution, and acceptance of the required notices. This step is used for access tracking and statistical use.

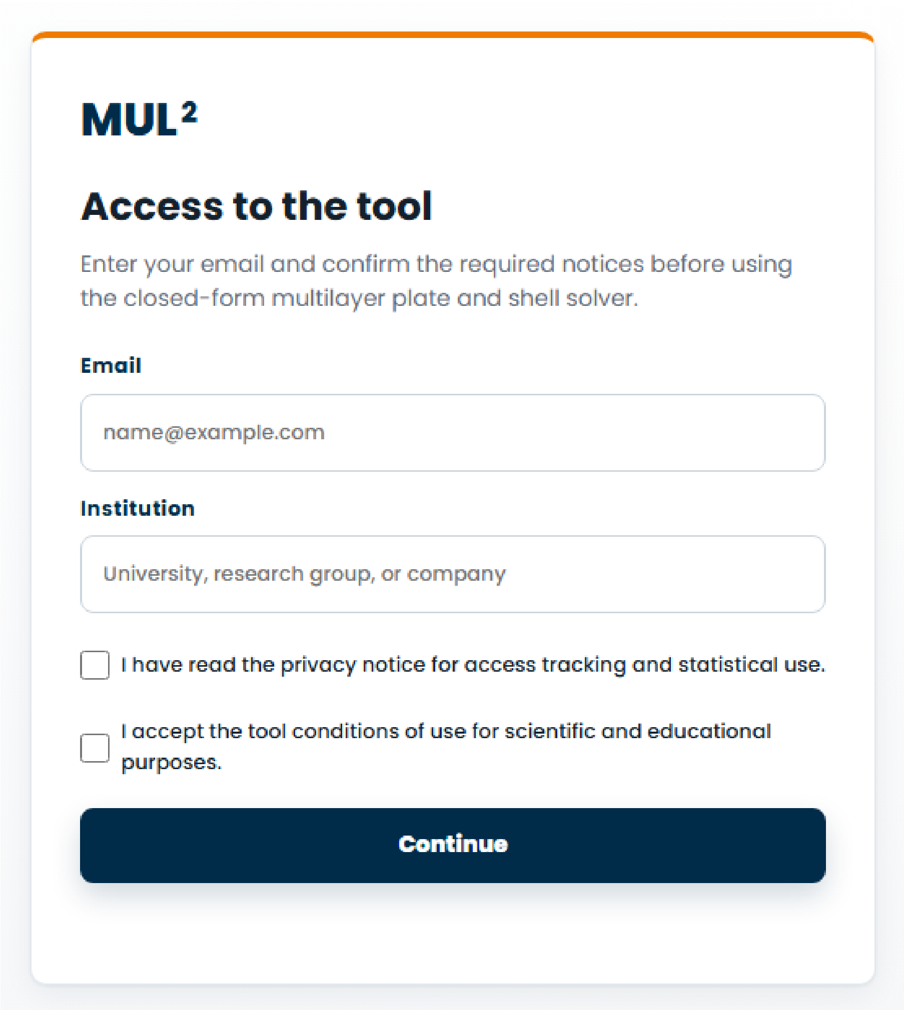


Figure 1: *Access screen shown before entering the solver.*

1. Enter a valid email address.
2. Enter the institution, research group, company, or other affiliation.
3. Select both checkboxes.
4. Press **Continue**.

If one field is missing, the form remains locked and reports the missing item. Once access is accepted, the solver opens in the same page.

3 Case Setup

The case panel is arranged from top to bottom in the same order used by the solver: units, analysis options, in-plane geometry, load, evaluation point, laminate definition, and optional custom materials.

3.1 Units and Shear Correction Options

The unit selector affects geometric quantities and displacement output. The current interface supports metres and millimetres for geometry. The shear-correction block controls optional post-processing and corrected FSDT shear correction computations.

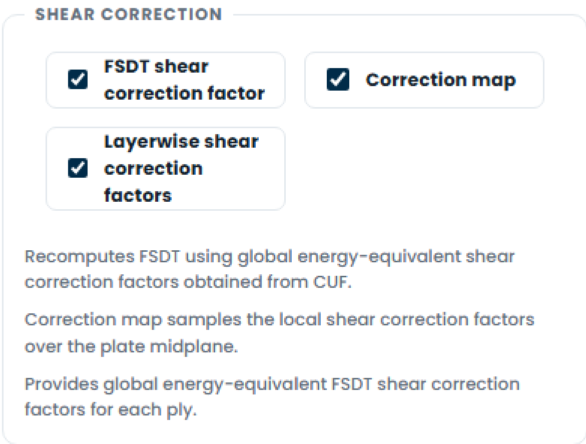


Figure 2: *Shear-correction controls. Secondary outputs become available after the main FSDT correction option is selected.*

The available shear-correction options are:

- **FSDT shear correction factor**: evaluates global transverse shear correction factors for the FSDT formulation commonly adopted in commercial finite element codes, using the exact analytical solution as reference. The corrected factors are then used to recompute the FSDT response.
- **Correction map**: evaluates the spatial distribution of the local shear correction factors over the plate midplane, providing a correction-factor map in the in-plane domain.
- **Layerwise shear correction factors**: reports ply-by-ply shear correction factors, highlighting how the correction varies through the laminate thickness and among the individual layers.

If the correction map or layerwise factors are not selectable, first enable **FSDT shear correction factor**.

3.2 Geometry

The geometry block defines the in-plane plate dimensions:

Input	Meaning
a	Plate length along the x-direction.
b	Plate length along the y-direction.

The total thickness h is not entered as an independent geometry input. It is computed automatically from the laminate stack,

$$h = \sum_{k=1}^{N_{\text{plies}}} t_k,$$

where t_k is the thickness of the k-th ply. The preview updates with a, b, the number of plies, and the individual ply thicknesses.

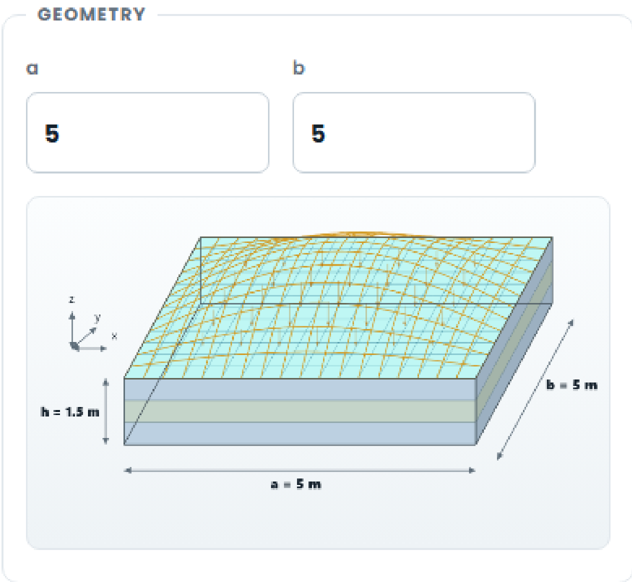


Figure 3: *Geometry block and laminate preview. Only a and b are entered in the geometry block; the total thickness h is derived from the laminate stack and shown in the drawing.*

The displayed value of h is the current sum of the ply thicknesses. Changing, adding, or removing plies immediately changes the total thickness used by the solver and by the preview.

3.3 Load Definition

The basic load block contains only the quantities needed in ordinary use:

- **Load type**: harmonic, uniform, triangular, bi-triangular, patch, or point load.
- **Magnitude**: pressure for distributed loads, force for point loads.
- **Side**: top or bottom application side.

  The supported load types are summarized below.

Load type	Meaning	Advanced controls
Harmonic	Single bi-sinusoidal pressure term.	Half-waves m and n.
Uniform	Constant pressure over the plate.	Fourier truncation M and N.
Triangular	Linear ramp p_0x/a.	Fourier truncation M and N.
Bi-triangular	Tent load with peak at the plate centre.	Fourier truncation M and N.
Patch	Rectangular loaded area.	Patch limits and Fourier truncation M, N.
Point	Concentrated load at a prescribed point.	Load position and Fourier truncation M, N.

The plate preview follows the selected load type and gives immediate visual feedback on the load distribution, patch position, or point-load position.

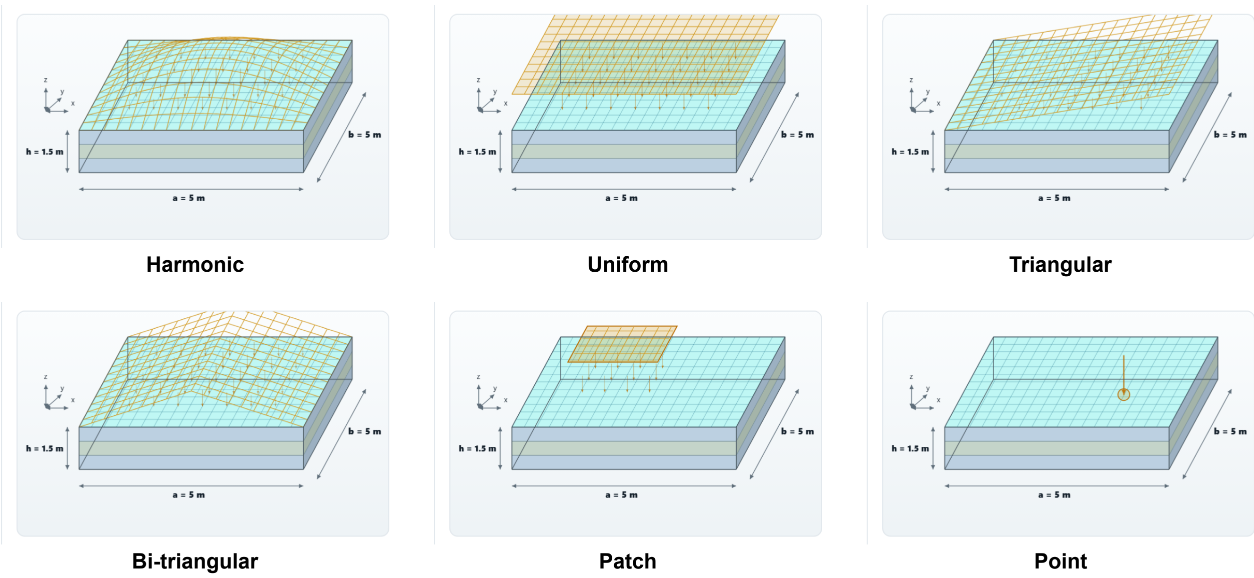


Figure 4: *Load preview for the supported transverse-load types. Patch and point previews update when their position parameters are changed in the advanced load settings.*

For a harmonic load, m and n select the number of half-waves along x and y.

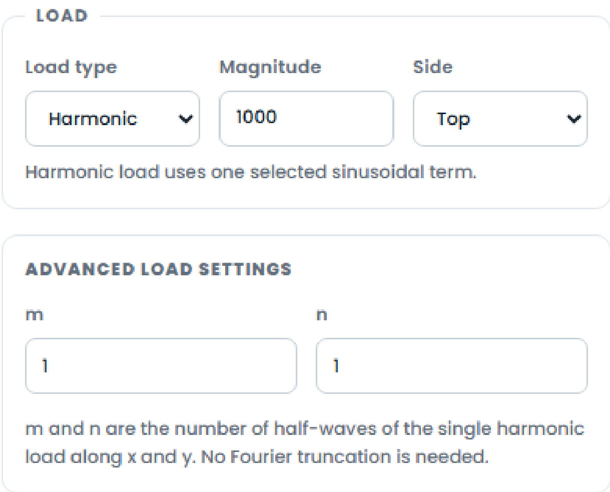


Figure 5: *Harmonic load settings. The advanced fields define the selected half-wave numbers m and n.*

For uniform, triangular, bi-triangular, patch, and point loads, the transverse load is represented by a truncated double sine series. The values M and N are the retained Fourier terms along x and y. Increasing them generally improves the reconstruction of the load and of the resulting stress field, but also increases the computation time.

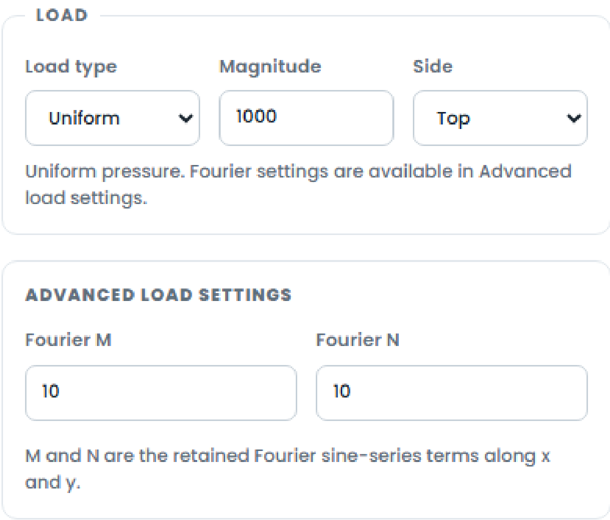


Figure 6: *Fourier settings for a load reconstructed through a sine-series expansion.*

Use M and N as convergence parameters. Smooth loads usually converge with moderate truncation. Patch and point loads contain stronger local variations; point loads are singular, so local stresses should always be interpreted through a convergence check.

3.4 Evaluation Point and Laminate Stack

The evaluation point is given through nondimensional coordinates x/a and y/b. It can be entered numerically or moved directly on the planform map.

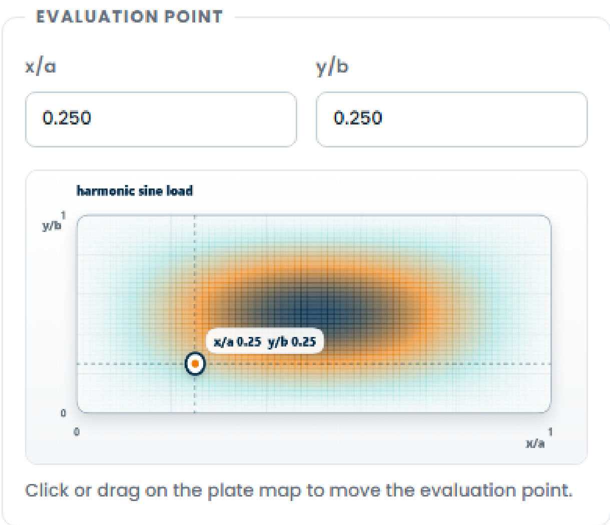


Figure 7: *Evaluation point selector. The point can be typed through x/a, y/b, or moved directly on the planform map.*

The laminate stack is entered one ply per row:

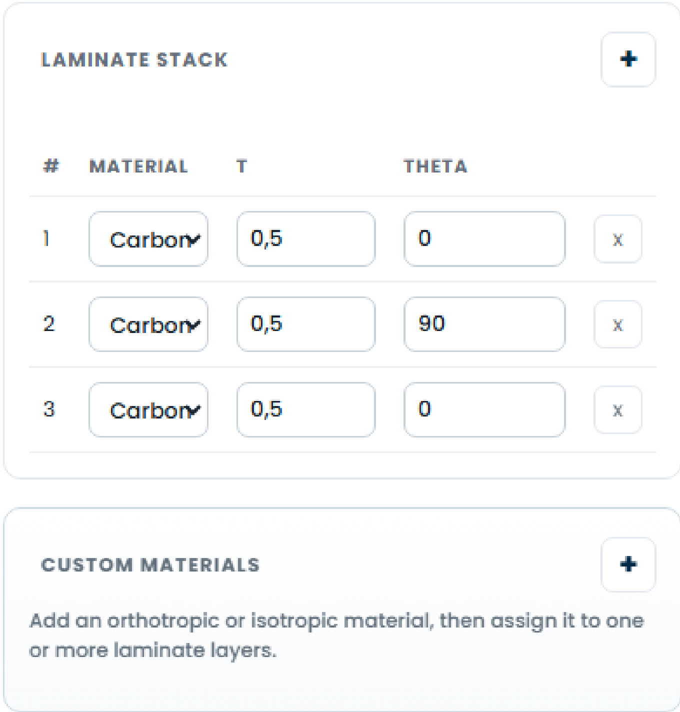


Figure 8: *Laminate stack and custom material panel. The total thickness used by the solver is obtained from the listed ply thicknesses.*

Column	Meaning
Material	Material assigned to the ply. Built-in and custom materials are listed in the same menu.
t	Ply thickness, in the selected geometry unit.
θ	Ply orientation angle in degrees.

The plus button adds a ply. The remove button deletes the corresponding row. The total laminate thickness is obtained directly from the ply table as $h = \sum t_k$.

3.5 Custom Materials

The custom material panel allows additional isotropic or orthotropic materials to be defined and reused in the laminate stack. Orthotropic materials require E_1, E_2, E_3, G_{12}, G_{13}, G_{23}, ν_{12}, ν_{13}, and ν_{23}. Isotropic materials require E and ν; the shear modulus is computed internally.

Scientific notation is accepted in the modulus fields, for example `1E+09`, `1E09`, `1e9`, or `1.25e+10`.

4 Running the Analysis

After completing the setup:

1. Check the ply thicknesses in the laminate stack; their sum defines h.
2. Check that the load type and side are correct.
3. Check the evaluation point on the planform map.
4. Press **Run Analysis**.

The solver returns to the results area when the computation is complete. For large Fourier truncation orders or concentrated loads, the computation can take longer because more terms are retained in the closed-form expansion.

If the run does not start, first check that every ply thickness is positive, that the access form is completed, and that the load parameters shown in the advanced load settings are valid.

5 Results

5.1 Summary Cards and Displacement Plots

The summary cards report the maximum absolute transverse displacement for each theory. The plot panel then shows the selected result group. The laminate overlay is enabled by default and displays ply regions behind the curves.

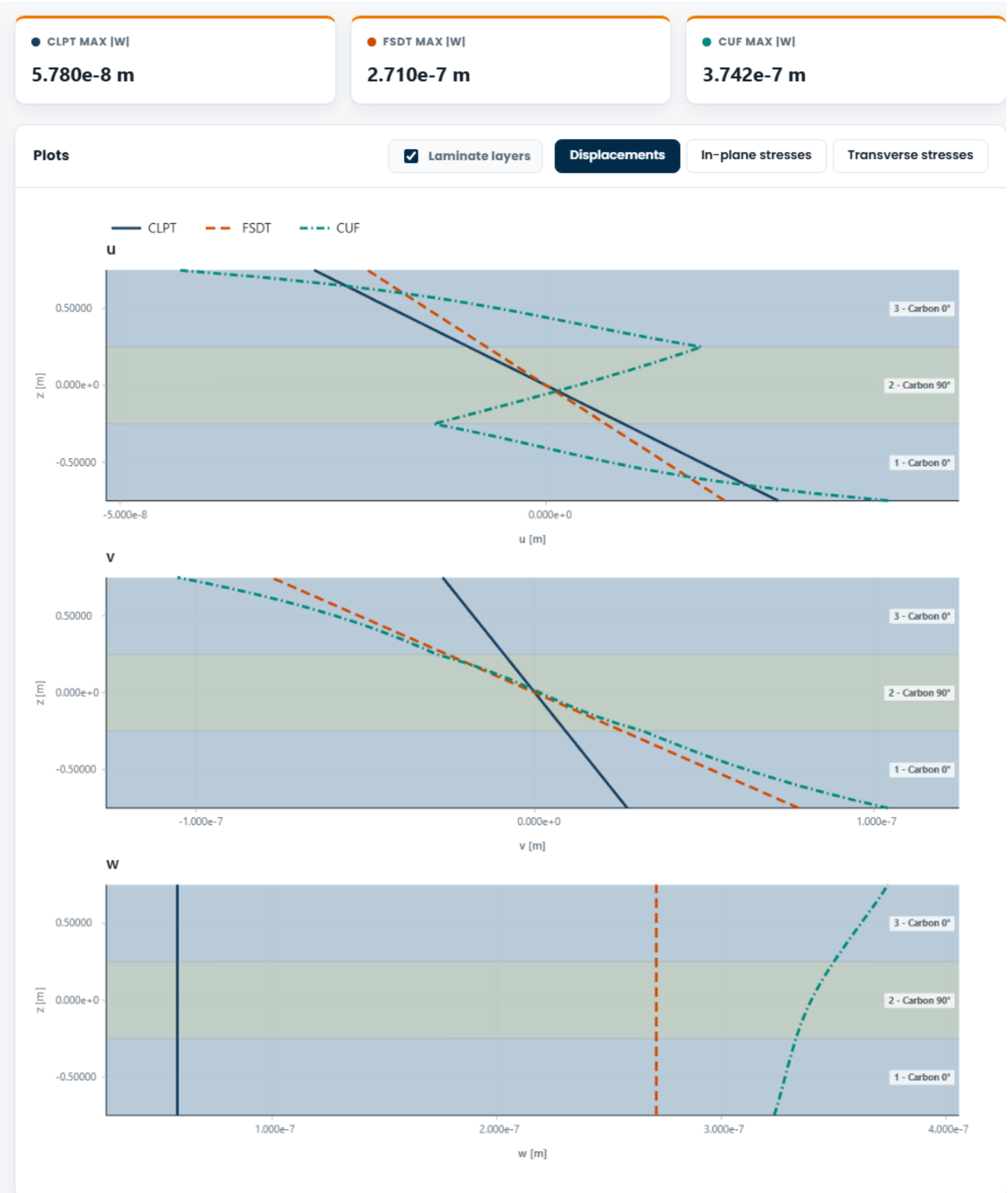


Figure 9: *Displacement plots with laminate overlay and theory comparison.*

The plot tabs are:

- **Displacements**: u, v, and w through the thickness.
- **In-plane stresses**: σ_{xx}, σ_{yy}, and σ_{xy}.

- **Transverse stresses**: σ_{xz}, σ_{yz}, and σ_{zz}.

The legend uses one curve for each displayed theory. CUF is shown as the refined solution used for comparison.

5.2 Transverse Stresses and Shear Correction Factors

The transverse-stress tab is the main view for inspecting shear stresses and recovered normal transverse stress. If shear-correction options were enabled before the run, additional collapsible panels appear below the main plot.

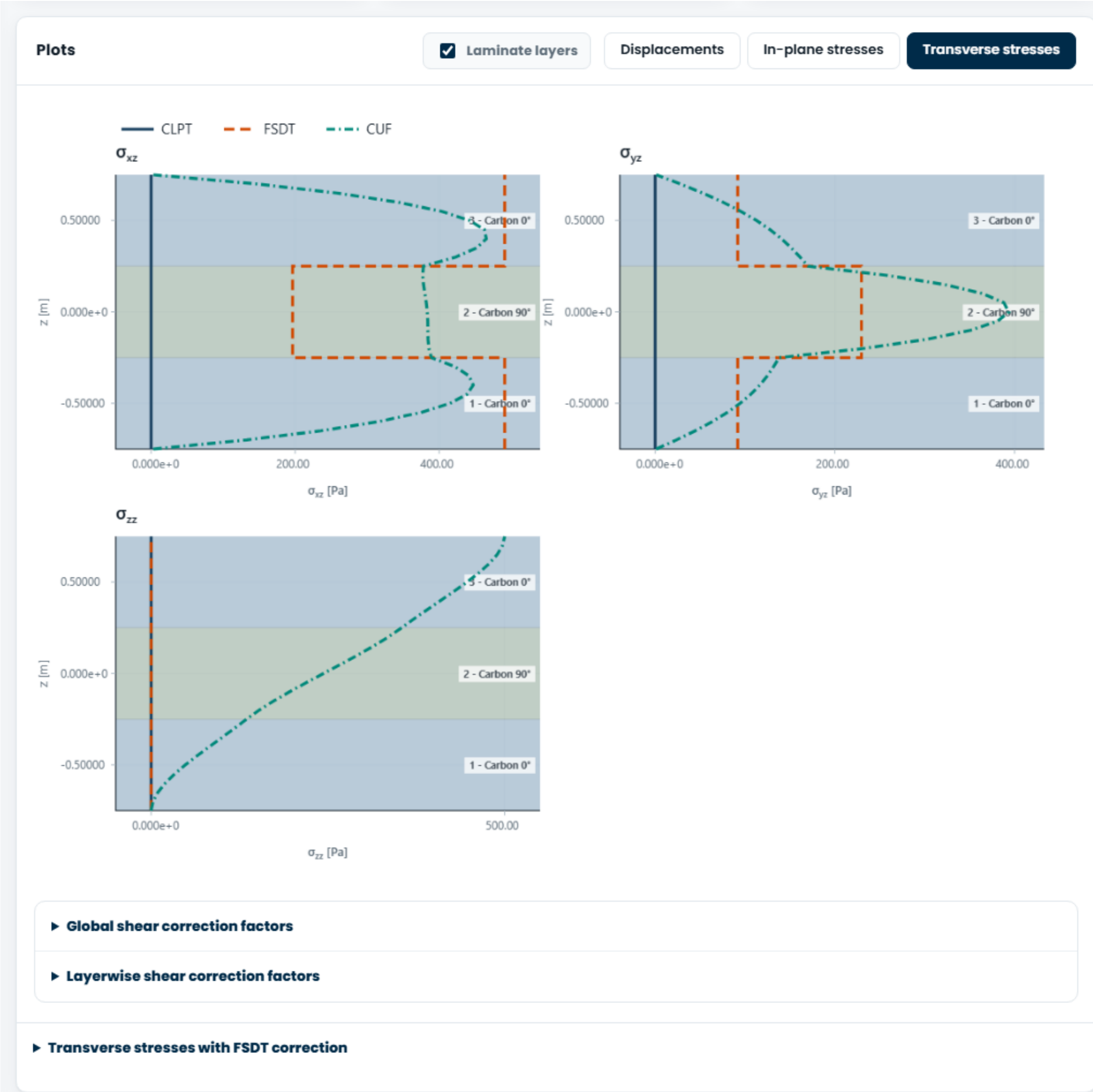


Figure 10: *Transverse-stress plot and collapsible shear-correction outputs.*

The shear-correction panels contain:

- global κ_{xz} and κ_{yz} factors for the plate;
- optional maps of local correction factors over the midplane;
- optional layerwise correction factors;
- corrected FSDT transverse-stress plots when the correction is active.

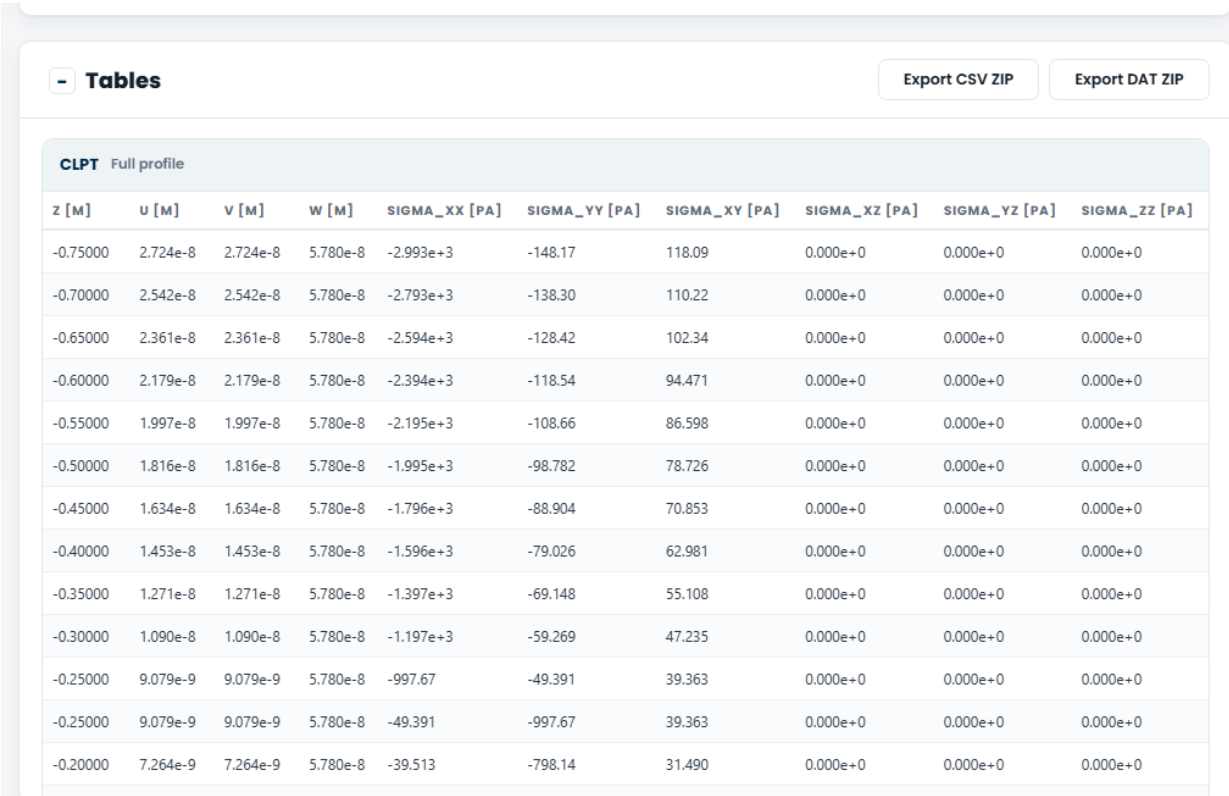
The global factors are displayed as two scalar values, one for each transverse-shear direction.

Figure 11: *Expanded global shear-correction factor panel. The reported values are the global κ_{xz} and κ_{yz} factors used for the corrected FSDT calculation.*

The global factors are intended for a corrected FSDT computation. The maps and layerwise factors are diagnostic outputs and help identify how the correction varies with position and laminate architecture.

5.3 Tables and Export

The table panel is collapsed by default to keep the result view compact. Open it when numerical values are needed.


- Tables   Export CSV ZIP   Export DAT ZIP

CLPT Full profile

Z [M]	U [M]	V [M]	W [M]	SIGMA_XX [PA]	SIGMA_YY [PA]	SIGMA_XY [PA]	SIGMA_XZ [PA]	SIGMA_YZ [PA]	SIGMA_ZZ [PA]
-0.75000	2.724e-8	2.724e-8	5.780e-8	-2.993e+3	-148.17	118.09	0.000e+0	0.000e+0	0.000e+0
-0.70000	2.542e-8	2.542e-8	5.780e-8	-2.793e+3	-138.30	110.22	0.000e+0	0.000e+0	0.000e+0
-0.65000	2.361e-8	2.361e-8	5.780e-8	-2.594e+3	-128.42	102.34	0.000e+0	0.000e+0	0.000e+0
-0.60000	2.179e-8	2.179e-8	5.780e-8	-2.394e+3	-118.54	94.471	0.000e+0	0.000e+0	0.000e+0
-0.55000	1.997e-8	1.997e-8	5.780e-8	-2.195e+3	-108.66	86.598	0.000e+0	0.000e+0	0.000e+0
-0.50000	1.816e-8	1.816e-8	5.780e-8	-1.995e+3	-98.782	78.726	0.000e+0	0.000e+0	0.000e+0
-0.45000	1.634e-8	1.634e-8	5.780e-8	-1.796e+3	-88.904	70.853	0.000e+0	0.000e+0	0.000e+0
-0.40000	1.453e-8	1.453e-8	5.780e-8	-1.596e+3	-79.026	62.981	0.000e+0	0.000e+0	0.000e+0
-0.35000	1.271e-8	1.271e-8	5.780e-8	-1.397e+3	-69.148	55.108	0.000e+0	0.000e+0	0.000e+0
-0.30000	1.090e-8	1.090e-8	5.780e-8	-1.197e+3	-59.269	47.235	0.000e+0	0.000e+0	0.000e+0
-0.25000	9.079e-9	9.079e-9	5.780e-8	-997.67	-49.391	39.363	0.000e+0	0.000e+0	0.000e+0
-0.25000	9.079e-9	9.079e-9	5.780e-8	-49.391	-997.67	39.363	0.000e+0	0.000e+0	0.000e+0
-0.20000	7.264e-9	7.264e-9	5.780e-8	-39.513	-798.14	31.490	0.000e+0	0.000e+0	0.000e+0

Figure 12: *Expanded table panel and export controls.*

Each table contains the through-thickness coordinate, the three displacement components, and the six stress components:

$$u,\quad v,\quad w,\quad \sigma_{xx},\quad \sigma_{yy},\quad \sigma_{xy},\quad \sigma_{xz},\quad \sigma_{yz},\quad \sigma_{zz}.$$

Two export formats are available:

- **Export CSV ZIP**: comma-separated files for spreadsheet or script use.
- **Export DAT ZIP**: aligned text tables with the same data content.

6 Units

The unit selector controls geometry input and displacement output. Material stiffness and stresses remain in SI stress units.

Quantity	When unit is `m`	When unit is `mm`
Geometry a, b, t **and derived** $h = \sum t_k$	metres	millimetres
Displacements in plots/tables/export	metres	millimetres
Material stiffness	Pa	Pa
Distributed load magnitude	Pa	Pa
Point load magnitude	N	N
Stresses in plots/tables/export	Pa	Pa

When millimetres are selected, do not convert material moduli to MPa unless that is explicitly intended. Built-in and custom material stiffness values are interpreted as Pa.

7 Recommended Workflow

For a new case, use the following order:

1. Select the length unit.
2. Enter the in-plane dimensions a and b.
3. Choose the load type, magnitude, and side.
4. Set the evaluation point.
5. Build the laminate from bottom to top; the total thickness is computed as $h = \sum t_k$.
6. Add custom materials only if the built-in list is not sufficient.
7. Enable shear-correction outputs only when they are needed.
8. Run the analysis.
9. Inspect plots first, then open tables or export files for numerical post-processing.