Composites in PrePoMax – User Guide

Overview

PrePoMax models laminated composite shells, solves them with CalculiX and checks every ply for failure. You define how strong the ply material is, which way the fibres run and how the plies are stacked; PrePoMax then shows stresses in the fibre directions and failure indices layer by layer.

A typical workflow:

1. **Material** — define the ply stiffness (Lamina) and its strength (Composite Strength).
2. **Material Orientation** — tell PrePoMax which direction is 0° on the shell.
3. **Composite Shell Section** — stack the layers: material, thickness and angle of each.
4. **Mesh** with parabolic shell elements (S8R quads, S6 triangles).
5. **Run** the analysis as usual.
6. **Results** — add field outputs for ply stresses and failure criteria, and view them per layer.

If you skip step 2, 0° is the global X axis projected onto the shell.

Defining the ply material

Use **Property → Material → Create** and add two properties to the ply material.

Elasticity → Lamina

Enter E1 (fibre direction), E2 (transverse), v12, G12, G13 and G23. PrePoMax completes the material as transversely isotropic (E3 = E2, v13 = v12, v23 = E2 / (2 G23) − 1) and refuses a G23 that would make it unstable. **Engineering Constants** (all nine values) also work if you have full 3D data.

Strength → Composite Strength

These values are used only by the failure criteria in the results; they are not sent to the solver. All strengths are positive, including the compressive ones.

Group	Field	Meaning
Fibre direction	Xt	Tensile strength along the fibres (axis 1)
Fibre direction	Xc	Compressive strength along the fibres
Transverse direction	Yt	Tensile strength across the fibres (axis 2)
Transverse direction	Yc	Compressive strength across the fibres
Shear	S12	In-plane shear strength
Shear	S23	Transverse shear strength; used by Hashin's matrix compression mode

Group	Field	Meaning
Interaction	Tsai-Wu F12*	Tsai-Wu interaction coefficient, between −1 and 1; default −0.5

All values accept expressions and parameters. Composite Strength has no temperature dependence. A layer whose material has no Composite Strength shows no failure value in the results.

For mass and inertia, give each ply material a **Density**: the part mass sums each layer with the density of its own material.

Fibre directions and the layup

Material Orientation (Property → Material Orientation → Create)

A material orientation sets the 0° direction (material axis 1) on a region of shell elements. It appears in the model tree under **Material Orientations**, between Materials and Sections.

Field	Options
Region type	Selection, Part, or Element set
Coordinate system	Global, or any rectangular or cylindrical coordinate system you created
Reference axis	X, Y or Z axis of that system — projected onto each element to become axis 1
Rotation angle	Extra turn of axis 1 about the shell normal, in degrees
Symbol density	How many axis symbols to draw, 0–100 % (display only)

Axis 3 always follows the shell normal, and axis 2 completes the right-handed set. While the dialog is open, or when you select the orientation in the tree, PrePoMax draws the axes 1, 2, 3 on the elements. A label `1*` means the chosen axis was almost along the normal (within 0.1°), so the Z axis (or X, if Z was chosen) was used instead. Use a cylindrical coordinate system for tubes and other curved parts. If two orientations cover the same elements, the later active one wins. Orientations can be deactivated, duplicated and renamed.

Composite Shell Section (Property → Section → Create)

Choose **Composite Shell Section** and select the region. The **Layers** tab holds one row per layer.

Column	Meaning
Layer	Layer number; layer 1 is on the **bottom**, opposite the shell normal
Material	Material of the layer
Thickness	Layer thickness; a unit may be typed with the value

Column	Meaning
Angle	Fibre angle in degrees, measured from axis 1 of the material orientation about the normal

Thickness and angle accept expressions and parameters (for example `=T` or `=A + 45`). A new row copies the last one, and rows can be pasted from a spreadsheet. The **Symmetric layup** check box at the top of the Layers tab appends the layers again in reverse order, so you only enter half of a symmetric layup.

The **Data** group on the Properties tab shows:

- **Material** — editable for a single layer; otherwise the shared material or "Mixed".
- **Total thickness** — the sum of all layers (editable only for a single, non-symmetric layer).
- **Offset** — shifts the laminate relative to the meshed surface, in units of the total thickness: 0 = the mesh is the mid-plane, 0.5 = the mesh is the top face (layers lie below it), −0.5 = the mesh is the bottom face.
- **Symmetric layup** — the same switch as the check box on the Layers tab.

Below the grid, a **layup diagram** draws every layer to scale, bottom to top, with the reference surface and axis 3. Mirrored layers of a symmetric layup are drawn as mirror images. The selected row is marked by a triangle in the margin, and its fibre axes are drawn on the model. Drag the splitter to resize the diagram.

Element type requirement

CalculiX accepts composite shells only on **S8R** (8-node quads) and **S6** (6-node triangles). When you press OK on another element type, PrePoMax asks:

- **Yes** — change the part's element types to S8R/S6 and save the section.
- **No** — save the section; it stays invalid until the element type is changed.
- **Cancel** — return to the dialog.

If the part is meshed with linear elements or not meshed yet, mesh it with parabolic shell elements first.

Running the analysis

Nothing changes in how you create steps, loads and the job. PrePoMax writes the layup to CalculiX automatically, computing the fibre directions of every element itself, so what you saw in the dialogs is exactly what the solver uses.

- Request **STRESS** (S) and, for the plane-stress option in the results, **strain** (TOSTRAIN or E) in the field outputs.
- CalculiX writes no reaction forces (RF) for composite elements. Check reactions on non-composite parts if you need them.
- **Supports and prescribed displacements:** CalculiX turns each layer into a thin solid and applies a boundary condition on a shell node to every node through the thickness. That pins the thickness there, so at and next to supports the plies are not in plane stress, and their stresses can differ from lamination theory by several percent. Read ply stresses away from supports, and when you

compare with a hand calculation, load the plate with forces or moments rather than prescribed displacements.
- Thickness, angle and strength values given as parameters can be changed and re-run in batch mode (`-p` or a `.pmp` file); the orientations follow.

Viewing composite results

Open the results with the model they were solved from. PrePoMax then knows the fibre axes of every element and every layer.

Layers in the results tree

CalculiX turns every layer into its own thin solid, so ply values are never averaged between plies. Each layer becomes a result part named `<part>_Layer-<n>`, grouped under the part's name in the results tree, and an element set `<section>_LAYER-<n>`. Use **Show Only**, **Show** and **Hide** to look at one ply at a time. Layer 1 is the bottom layer; the mirrored layers of a symmetric layup continue the numbering upward.

Result field outputs (Results → Field Output → Create)

Type	What you choose	What you get
Material Axes	Any tensor field, e.g. STRESS	That field rotated into the fibre axes: component 11 along the fibres, 22 across them, 33 through the thickness
Shell In-Plane State	Quantity, then the source field	In-plane components (S11, S22, S12 or E11, E22, E12), in-plane principal values, maximum in-plane shear, and ANGLE from the fibre direction to the max principal direction (°)
Composite Failure	Criterion, Stresses, source field	FAILURE_INDEX, RESERVE_FACTOR, MODE, MAX_FAILURE_INDEX, CRITICAL_LAYER

Quantity / Stresses options:

- **Stress (3D as solved)** — the stresses CalculiX computed. Choose a stress field.
- **Stress (plane stress from strains)** — stresses recomputed from the in-plane strains with the ply's E1, E2, v12 and G12, as classical lamination theory does. Choose a strain field. Useful for comparing with hand calculations; the 3D transverse stress can differ by several percent.
- **Strain (in-plane)** — the in-plane strains (Shell In-Plane State only).

Failure criteria

Criterion	Character
Maximum stress	Each stress compared with its strength separately; reports the governing mode

Criterion	Character
Tsai-Hill	Single interactive quadratic index
Tsai-Wu	Interactive index with tension/compression difference and F12*
Hashin	Separate fibre and matrix modes (1980, plane stress); reports the governing mode

How to read the components:

- **FAILURE_INDEX** — 1.0 or more means the ply fails.
- **RESERVE_FACTOR** — how many times the load could be increased before failure (1 / index for maximum stress). Below 1 means failure.
- **MODE** — 1 fibre tension, 2 fibre compression, 3 matrix tension, 4 matrix compression, 5 in-plane shear; 0 for Tsai-Hill and Tsai-Wu.
- **MAX_FAILURE_INDEX** — the worst index through the whole thickness, shown on every layer.
- **CRITICAL_LAYER** — the layer where that worst index occurs.

Plies whose material has no Composite Strength, and non-shell regions, show no value.

Tensor plots

With **Vector plot** on the toolbar, a component that has a direction is drawn as a tensor plot: arrows along the axes it acts in.

Field	Drawn as a tensor plot	Arrows
Material Axes	S11, S22, S33	Along material axes 1, 2 and 3 of each layer
Material Axes	Principal values	Along the principal directions
Shell In-Plane State	S11, S22	Along axes 1 and 2, in the shell plane
Shell In-Plane State	PRINCIPAL_MAX, PRINCIPAL_MIN	Along the in-plane principal directions

Components without a direction (S12, SHEAR_MAX, ANGLE and all Composite Failure components) are drawn as a normal plot.

Limitations

- Composite layups work on **shell** parts only, meshed with **S8R or S6** elements.
- Failure criteria are in-plane (plane stress) criteria; no delamination, Puck, LaRC or progressive damage.
- Every layer covers the whole section region (no ply drop-offs); no draping simulation.
- Composite Strength is not temperature dependent.
- Importing composite sections or orientations from an `.inp` file is not supported.
- No reaction forces on composite elements (CalculiX).
- Fibre axes are previewed on the mesh only, not on the geometry before meshing.