Simcenter 3D Laminate Composites

Modeling and simulating composite structures

Benefits
- Reduce laminate model creation time by choosing between zone-based modeling, ply-based modeling or a mixture of both approaches
- Leverage Simcenter 3D's open solver architecture to perform state-of-the-art dynamic, nonlinear, progressive failure and delamination simulations
- Keep your model up-to-date with the latest design through geometry associativity
- Interact with CAD-based composites definitions from Fibersim™ software, CATIA and others
- Use NX standard materials, or create ply materials from the constituent fiber and matrix material properties to simulate plies made of woven, unidirectional, randomly oriented short fibers, particulates, and to represent cores
- Conveniently assign laminates and plies to your choice of geometry, meshes and/or elements
- Improve finite element modeling accuracy by accounting for distorted fiber orientations
- Post-processing tools allow you to quickly identify critical plies and load cases using classical and user-defined failure theories, and to create reports

Summary
Simcenter™ 3D Laminate Composites software is a modular Simcenter 3D simulation toolset for laminate composite structures. Easy-to-use ply and laminate definition tools enable you to quickly create finite element models representing your laminate composite design. Simcenter 3D Laminate Composites helps you create, optimize and validate composite structures using NX™ Nastran®, LMS Samcef™, MSC Nastran, ANSYS, Abaqus or LS-DYNA as your solver. Laminate Post Reporting generates graphical and spreadsheet ply results from shell stress resultants and envelopes ply stresses, strains and failure metrics on elements and over multiple loads cases.

Zone-based modeling
Simcenter 3D Laminate Composites enables you to attach your laminates directly to 2D or 3D mesh collectors. An interactive graphical modeler helps you to create and stack laminated composite plies efficiently. You can use standard pre-defined stacking sequences and/or ply groups to minimize the effort required to create and modify complex, multi-ply laminates.

You can quickly define complex layups by importing them from Microsoft Excel or by using shorthand notation.

Ply-based modeling
You can use Simcenter 3D Laminate Composites to create global plies, assign them to polygon faces and/or 2D elements and define ply orientations by projection of the material orientation or by using one of the draping algorithms. Simcenter 3D automatically computes element zones for solver export. You can assign different layup offsets on different faces and view the fiber orientations on the 2D meshes.
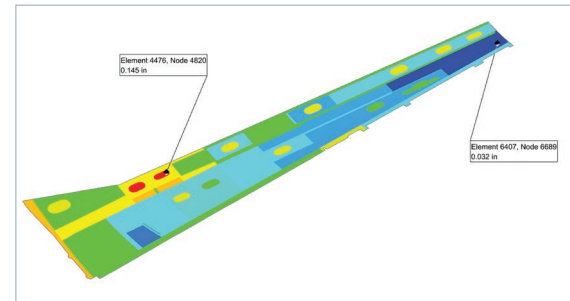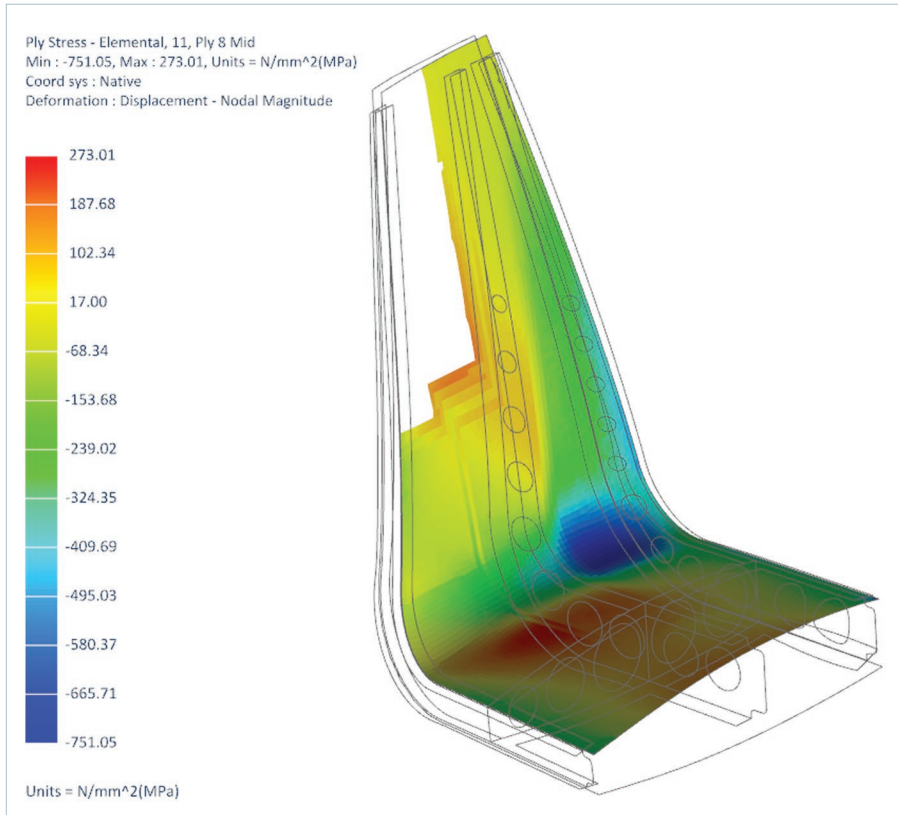

Fibersim interaction
Use the bi-directional Fibersim ply exchange to communicate with composites designers. Plies can be imported directly from the Simcenter .prt file, or via an HDF5 file. You can also exchange zone and laminate definitions via the zone interfaces.

Interface with other tools
Import ply definitions from third-party tools such as CATIA, Patran Laminate Modeler and Simulayt Laminate Tools.

www.siemens.com/plm/simcenter3d

Simcenter 3D Laminate Composites

Ply Stress - Elemental, 11, Ply 8 Mid
Min : -751.05, Max : 273.01, Units = N/mm^2(MPa)
Coord sys : Native
Deformation : Displacement - Nodal Magnitude

273.01
187.68
102.34
17.00
-68.34
-153.68
-239.02
-324.35
-409.69
-495.03
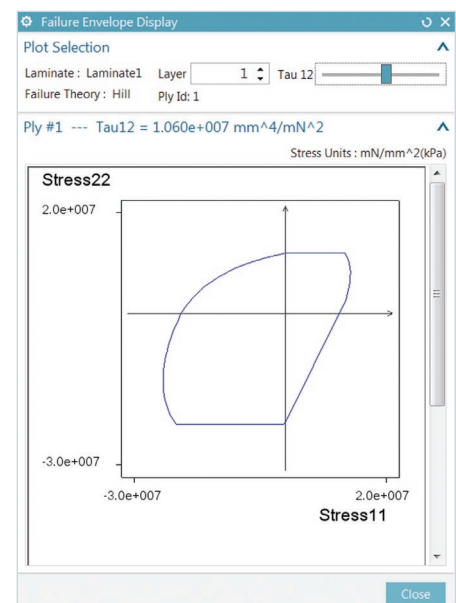-580.37
-665.71
-751.05

Units = N/mm^2(MPa)

Draping

Simcenter 3D Laminate Composites draping algorithms predict the fiber orientations and shear of unidirectional and woven plies as they drape around doubly-curved or undevelopable surfaces. Simcenter 3D enables you to specify a draping start point and an initial draping direction, or to pick a seed curve. You can then coarsen or refine the drape mesh size and view the ply flat pattern.
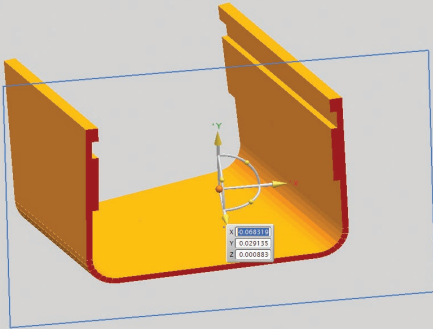
Ply materials

You can combine fiber and matrix properties to create ply materials, which can be used in the laminate stack just like standard Simcenter 3D materials. Micromechanics theories allow you to create ply materials with the following types of fibers:

• Unidirectional

• Woven (you can specify the angle between the warp and weft fibers)

• Particulate

• Randomly oriented short fiber

A core ply material allows you to define thickness-dependent shear stress limits.

3D layup inflation

Simcenter Laminate Composites' 3D Inflation tool automatically creates detailed, solid laminate models that were previously too tedious to attempt. You can extrude normal to a 2D mesh or by filling the volume between 2D dependent meshes. You can taper core or filler plies. A dedicated sandwich panel capability allows the automatic generation of 2D or 3D facesheet plies along with a user-defined number of solid element layers for the core. Cohesive elements can be introduced between selected plies, for assessment of delamination using Samcef.

Dynamic simulation

Use NX Nastran, Simcenter 3D Response Dynamics and the Simcenter 3D Laminate Composites dynamics metasolution to generate ply results for dynamic events. The dynamics metasolution uses advanced algorithms and a parallelized solver to efficiently compute ply stresses, strains, failure indices, strength ratios and margins of safety for base-driven random and harmonic events.

Failure Envelope Display

Plot Selection

Laminate : Laminate1   Layer   1   Tau 12
Failure Theory : Hill   Ply Id: 1

Ply #1 --- Tau12 = 1.060e+007 mm^4/mN^2

Stress Units : mN/mm^2(kPa)

Stress22

2.0e+007

-3.0e+007

-3.0e+007   2.0e+007

Stress11

Close

Laminate validation

To visually validate your laminate, you can view the thickness and the offsets (both nodal and reference plane location) of your laminate physical properties and your layups. Contour plots of laminate thickness are also possible, helping you visually understand your model.

You also can view the element zones and assigned laminate property, as well as create element groups from those zones to more easily manage your FE model. Laminate stiffness matrices (A, B, D and S) can be displayed in both Microsoft Excel and text formats. Similarly, equivalent engineering constants can also be displayed, enabling you to assess the characteristics of your laminate prior to solution.

You can apply loads and temperatures to your laminate and review the ply stresses, strains, failure indices and margins of safety.

Failure theories

Simcenter 3D Laminate Composites computes failure metrics from ply stresses and strains, using the following failure theories:

- Maximum stress
- Maximum strain
- Hill
- Hoffman
- Tsai-Wu
- Puck
- LaRC02
- Von Mises
- Core shear
- User-defined first ply failure
- User-defined progressive failure with Abaqus

Optimization

Simcenter 3D laminate optimization allows you to improve the fundamental properties of your laminate. You can optimize a laminate physical property to meet stiffness, stability, strength or mass objectives. A genetic optimizer handles continuous variables such as an orientation angle and ply thickness, as well as discrete variables such as the existence of a ply, ply material, thickness and orientation. The software provides five laminate definitions that come the closest to meeting the objectives. You can select one of the five to replace the original laminate.

In addition, your continuous thickness and orientation design variables can be exported as modeling objects and used to define NX Nastran Solution 200 design optimization studies.
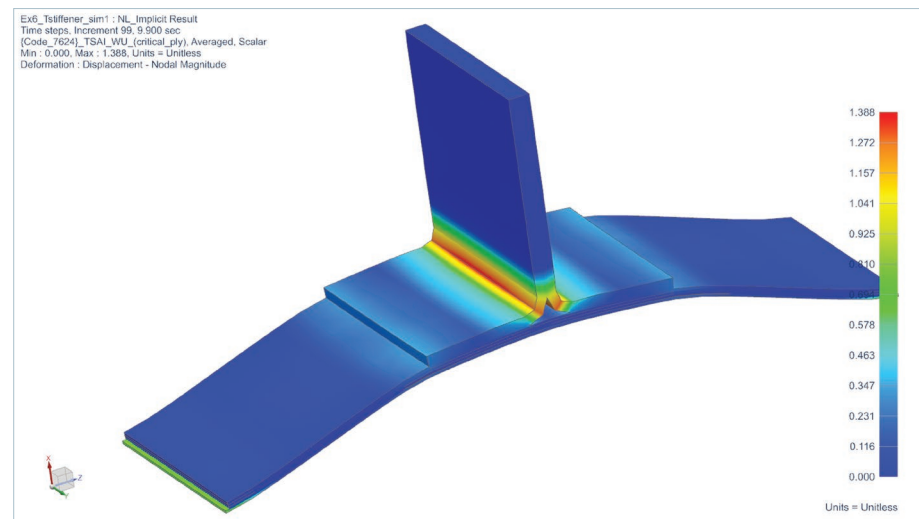
Solver support

You can assign laminates to shell or solid elements, which then can be used for the following supported solvers.

Nastran
- PCOMP/PCOMPG/PSHELL
- PCOMPS/PSOLID
- 2D: CQUAD4/R/8, CTRIA3/R/6
- 3D: CHEXA, CPENTA

Samcef
- 2D: Mindlin Shell (T28, T29), Heterosis Shell (T155,T156), Membrane (T57, T58), Membrane Shell (T53), Thermal membrane (T34, T35)
- 3D: Solid (T8, T46), Solid Shell (T153, T154), Cohesive Element (T145, T147), Non local element (T182, T183), Thermal solid (T039)

ANSYS
- 2D: SHELL181, SHELL281, SHELL99, SHELL91, SOLSH190
- 3D: SOLID186, SOLID191, SOLSH190

Abaqus
- 2D: S8R, S8R5, S4, S4R, S4R5, STRI65, S3, S3R, STRI3
- 3D: SC6R, SC8R, C3D8, C3D8H, C3D8R, C3D8RH, C3D8I, C3D8IH, C3D20, C3D20H, C3D20R and C3D20RH

LS-DYNA
- 2D: ELEMENT_SHELL, PART_COMPOSITE
- 3D: ELEMENT_TSHELL

GEOMETRIC SOLUTIONS

Solution Partner	SIEMENS	Gold Smart Expert
PLM		Channel

844-GEO-SUPT
support@geoplm.com
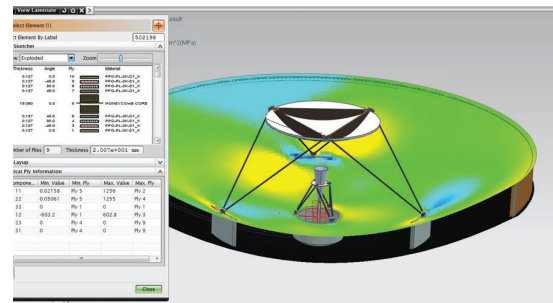geoplm.com

Postprocessing

Simcenter 3D postprocessing allows you to view shell stress resultants, ply stresses, ply strains, ply failure indices (ply and bond) and ply strength ratios (ply and bond).

Simcenter 3D Laminate Composites provides enveloping, filtering and sorting tools that enable you to quickly identify the critical element and ply over many solutions, including imported results. Simcenter 3D reads solver shell stress resultants and/or ply stresses, and computes ply strains, stresses, ply failure indices and margins of safety. It then graphically displays the critical ply ID for each element.

Simcenter 3D Laminate Composites also computes core ply transverse shear failure indices, strength ratios and margins of safety using allowable shear stresses in the 23 and 13 directions.

Simcenter 3D graphical post reporting uses the laminate properties in your active FEM, so that you can switch failure theories and even modify your laminates after the solution to perform what-if analyses. Reporting will update draping and zones if required, prior to accessing results.

To speed reporting for multiple or large solution files, you can use Quick Post reporting, which does not query the active FE model.