Introduction to the Autodesk® Simulation Composite Products

Doug Kenik – Autodesk, Inc.
Dan Milligan – Autodesk, Inc.

SM1830

In this class, we provide a broad overview of the newly released composites software functionality. We focus on Autodesk® Simulation Composite Design software for designing composite structures and on Autodesk® Simulation Composite Analysis software for stress and failure analysis of composite structures. We provide an overview of the capabilities of the software, present case studies, and discuss target markets. We also cover the future direction of the Simulation Composites product line.

Learning Objectives

At the end of this class, you will be able to:

- List and describe the capabilities of Autodesk Simulation Composite Design
- List and describe the capabilities of Autodesk Simulation Composite Analysis
- Use Autodesk software to design and analyze composite structures
- Simulate composite materials

About the Speakers

Doug Kenik was born and raised in Wyoming and attended college at the University of Wyoming where he pursued a Master's Degree in Mechanical Engineering studying micromechanics of composite materials. Directly after obtaining his Master's degree, he was employed by Firehole Technologies as a software developer and researcher. Doug spent 5 years at Firehole, the last 2 of which were directed at account management and product development. Firehole was eventually acquired by Autodesk, and now Doug is a Product Manager for the composites software which was acquired from Firehole.

Dan Milligan also came to Autodesk as part of the Firehole acquisition. He was a Composites Engineering Consultant during his 6 years at Firehole and is now a Technical Specialist at Autodesk where he helps users adopt and best utilize the composites software provided by Autodesk. Dan has developed an expertise in using simulation to analyze composite structures and training others on how to get the best results from their composites simulation. He has published papers and presented at numerous composites conferences and has taught several composites simulation and design courses.

Introduction

Increasing use of composite materials in a wide variety of industries has led, not surprisingly, to an increased demand for simulation of the various materials systems. The complex nature of composite materials has led for a high demand of simulation tools to cover a broad spectrum of topics, including material design, structural design and optimization, manufacturing, and advanced simulation/analysis.

Recently, Autodesk has taken a necessary step into the world of composite simulation by acquiring the technologies previously offered by Firehole Composites. In this class, we will cover the new composite simulation tools offered by Autodesk, provide examples for the tools, and discuss the future of composite material simulation.

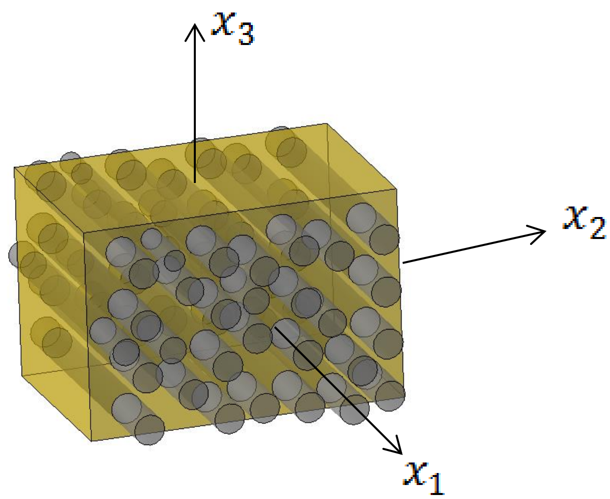
Overview of Composite Materials

Composite materials are materials which are comprised of two or more materials with clearly different material properties. Examples of composite materials include:

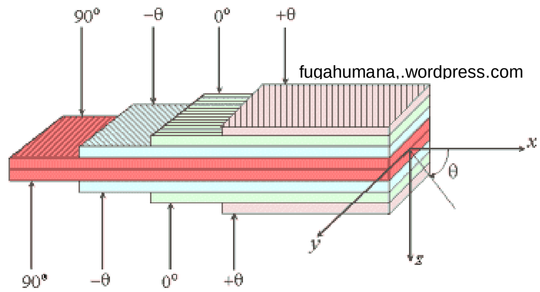
- Wood
- Polymer matrix composites (PMC)
  - Chopped fiber
  - Continuous fiber
- Ceramic matrix composites (CMC)
- Metal matrix composites (MMC)
- Concrete

The materials which jointly make up the composite are called constituents. Each of the composite materials above can be called a lamina (also called a ply).

NOTE: The material properties of a composite ply are *typically* different depending on the axis of interest. For example, a continuous fiber unidirectional composite ply is typically stiffer and stronger in the x_1 direction than the x_2 and x_3 directions.



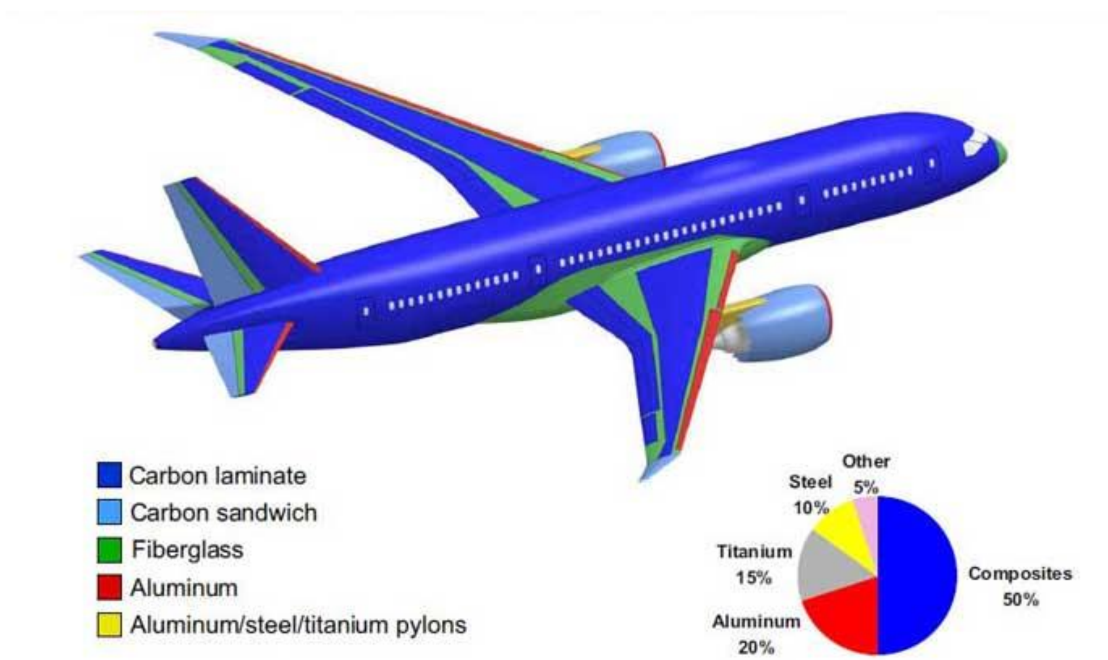
Depending on the manufacturing method, individual laminae (plural form of lamina) can be ‘stacked’ and oriented about a global axis to form a laminate.



$$[\theta, 0, -\theta, 90]_s$$

Again, the material properties of the laminate can be tailored to exhibit different responses depending on the axis of loading. Various materials can be inserted in a laminate, including metal plies, honeycomb cores, foam cores, etc.

Finally, various laminates (or lamina) can be combined to build a composite structure. The structure could potentially have ply drops, ply ramp ups, build up, etc. All of these include adding or subtracting material in areas where support is either needed or not required. As is typical with composite structures, weight savings and material costs dictate the positions where additional material can be saved.

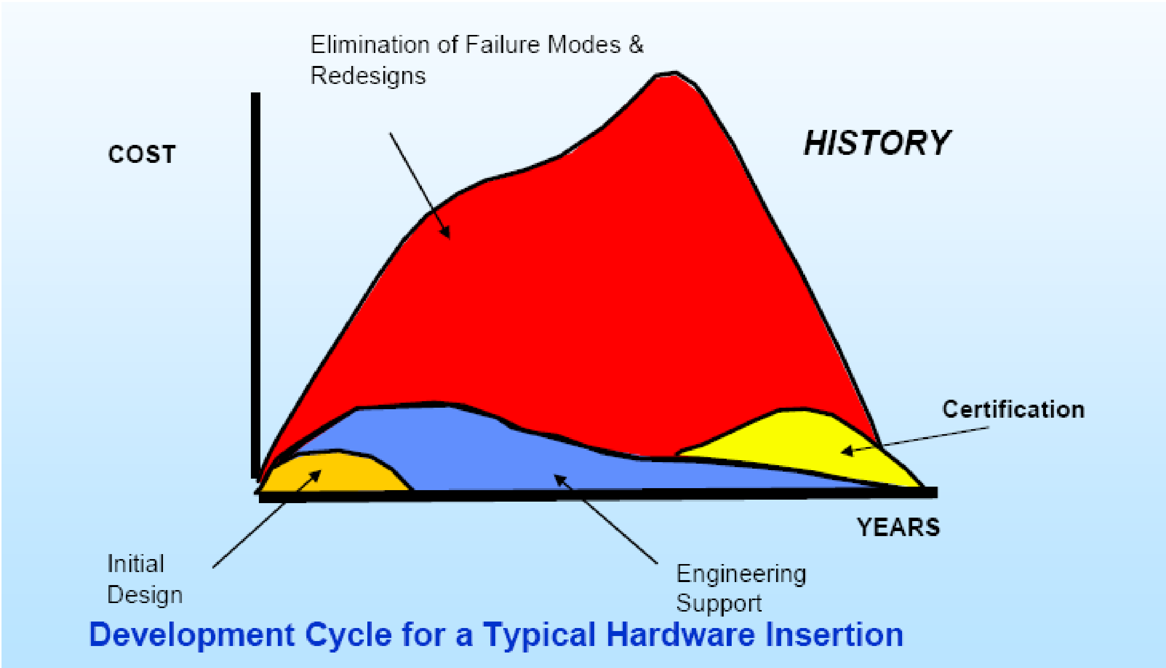


www.1001crash.com

Simulation of Composite Materials

The widespread adoption of composite materials has followed a typical cycle:

1. Materials are developed which exhibit superior performance
2. These materials are placed into production
3. Lack of simulation forces multiple testing cycles
4. The need for simulation is driven by the costs of testing



Courtesy Boeing Phantom Works

Simulation of composite materials falls into one of three interests:

1. Design
2. Manufacturing
3. Detailed analysis

Autodesk® Simulation Composite Design

What is it?

Autodesk Simulation Composite Design is a desktop application used to aid in the design of composite laminates for structural applications.

What does it do?

The functionality of Composite Design allows users to:

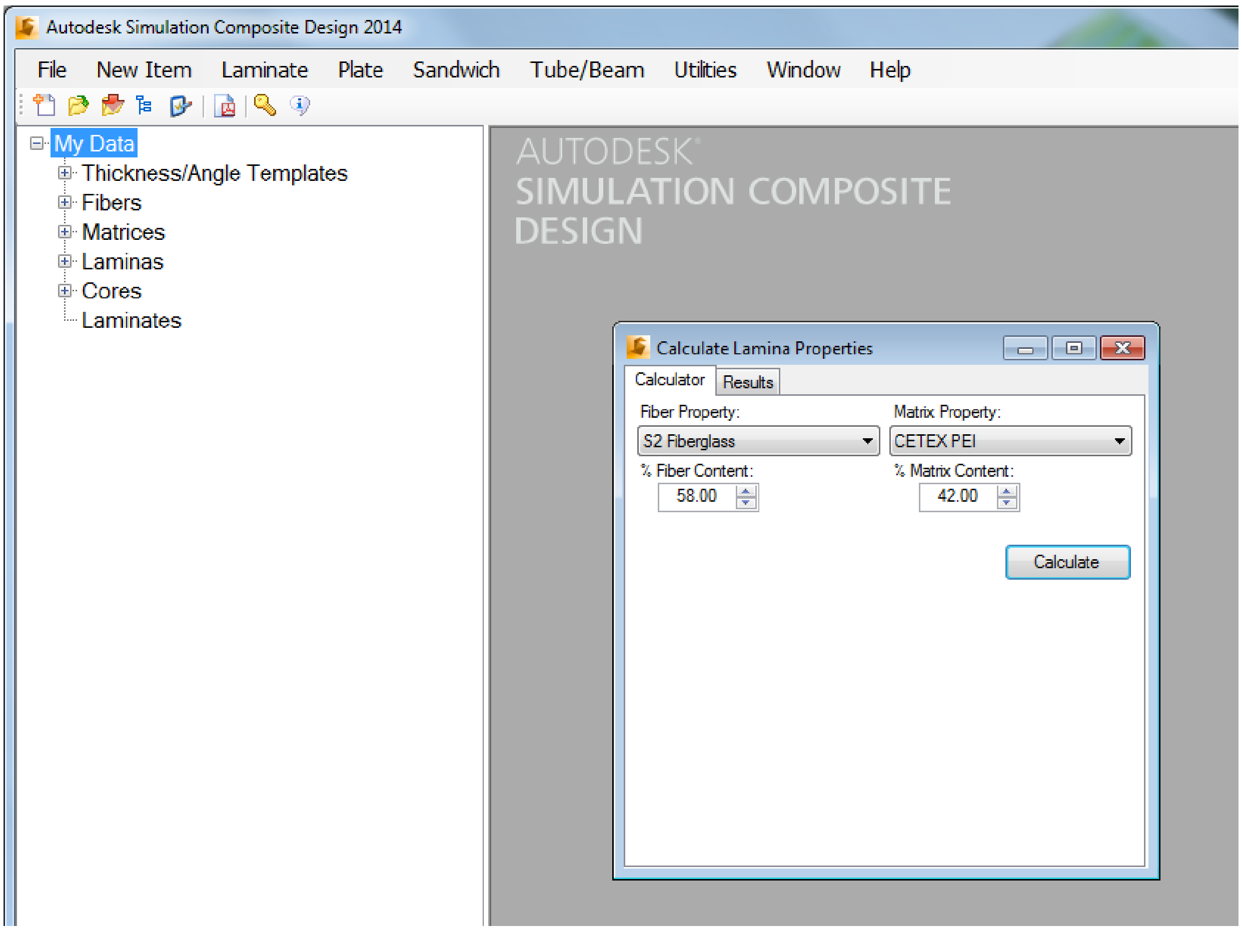
- Create lamina from micromechanics using individual fibers and matrices
- Create laminates using predefined laminae

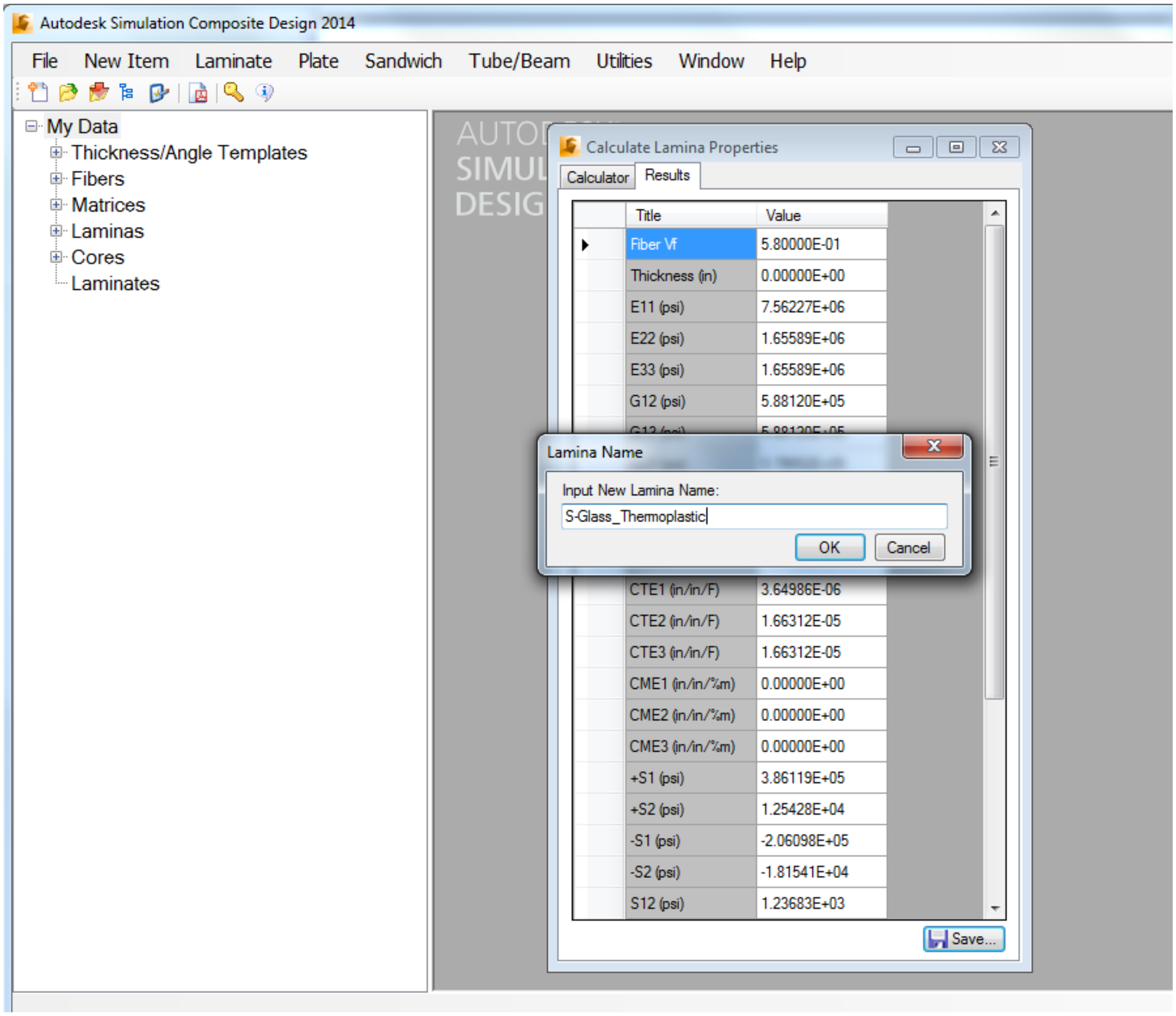
- Perform laminate analysis
    - Determine 3D properties and export to FEA codes
    - Determine the strengths of composite laminates
    - Plot failure envelopes
    - Perform progressive failure analyses
- Perform tube/beam analyses
    - Select from a wide variety of cross sections
    - Bending analysis
    - Stability analysis
    - Vibration analysis
- Perform sandwich analysis
    - Bending
    - Stability
    - Vibration
- Perform plate analysis
    - Bending
    - Stability
    - Vibration
- Perform pressure vessel analysis
    - Thin & thick wall
    - Open & closed end solutions
    - Failure analysis

Closed form solutions allow for on-the-fly simulations to aid in designing laminates for the application of interest.

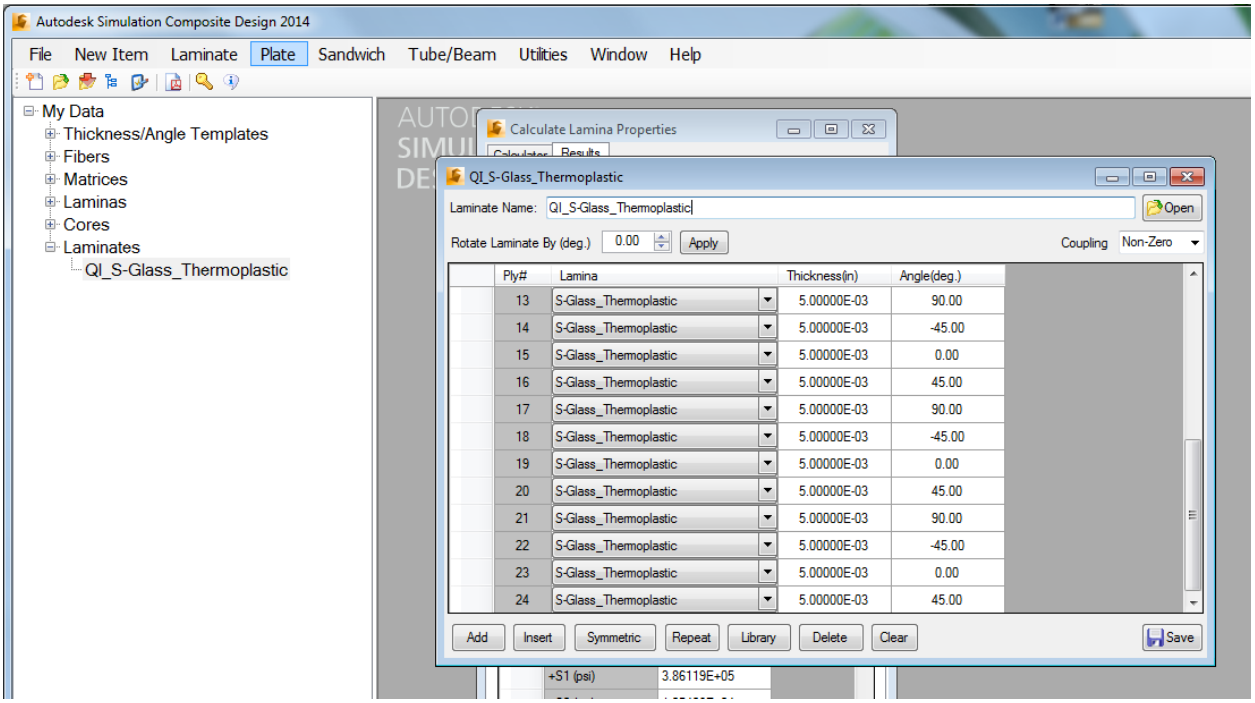
Example 1 – Create a Composite Laminate and Determine the Stiffness and Strength of the Laminate

For this example, we will create a 24 ply quasi-isotropic laminate made from 58% Fiber-Volume-Fraction S-Glass/Thermoplastic unidirectional composite plies. First, we can create a new ply (lamina) material using the S2 Fiberglass fiber and CETEX PEI thermoplastic matrix that come installed with the Autodesk Simulation Composite Design material library.

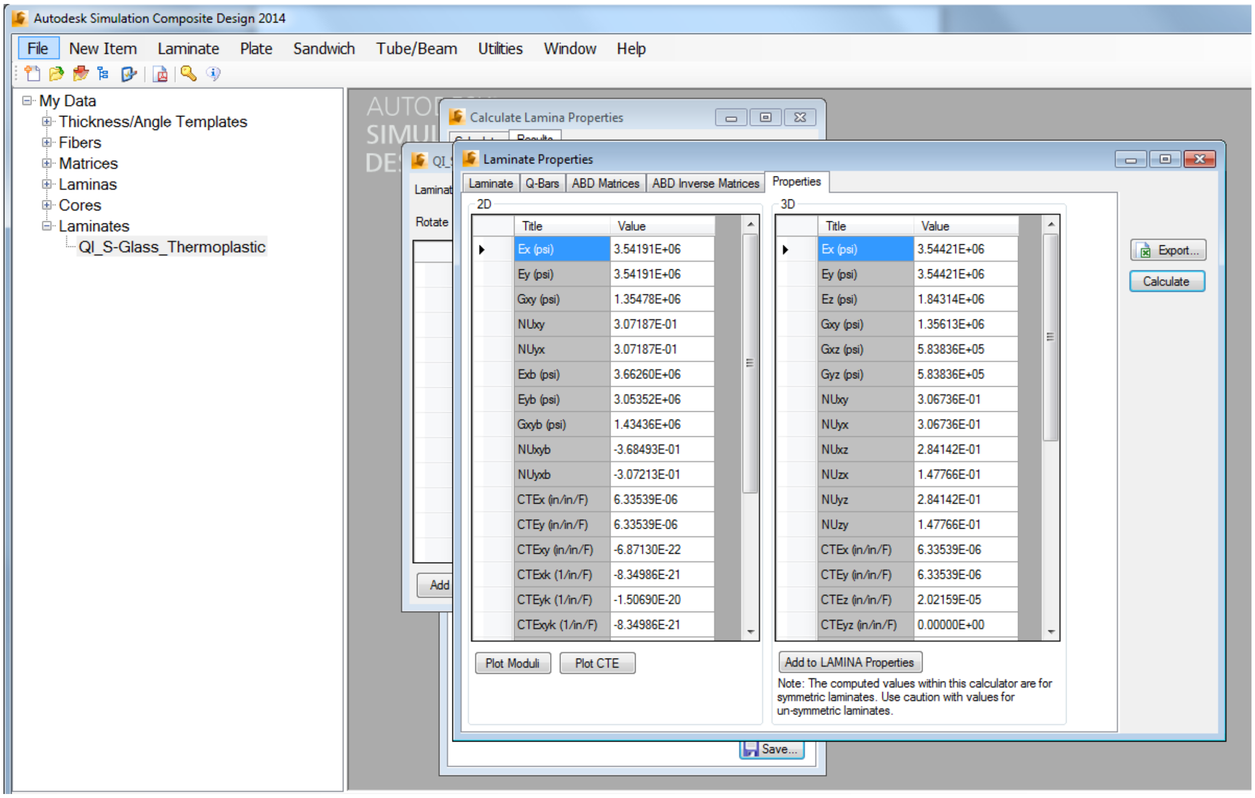



Autodesk Simulation Composite Design 2014
File
New Item
Laminate
Plate
Sandwich
Tube/Beam
Utilities
Window
Help
My Data
Thickness/Angle Templates
Fibers
Matrices
Laminas
Cores
Laminates
Calculate Lamina Properties
Calculator
Results
Title
Value
Fiber Vf
5.80000E-01
Thickness (in)
0.00000E+00
E11 (psi)
7.56227E+06
E22 (psi)
1.65589E+06
E33 (psi)
1.65589E+06
G12 (psi)
5.88120E+05
Lamina Name
Input New Lamina Name:
S-Glass_Thermoplastic
OK
Cancel
CTE1 (in/in/F)
3.64986E-06
CTE2 (in/in/F)
1.66312E-05
CTE3 (in/in/F)
1.66312E-05
CME1 (in/in/%m)
0.00000E+00
CME2 (in/in/%m)
0.00000E+00
CME3 (in/in/%m)
0.00000E+00
+S1 (psi)
3.86119E+05
+S2 (psi)
1.25428E+04
-S1 (psi)
-2.06098E+05
-S2 (psi)
-1.81541E+04
S12 (psi)
1.23683E+03
Save...

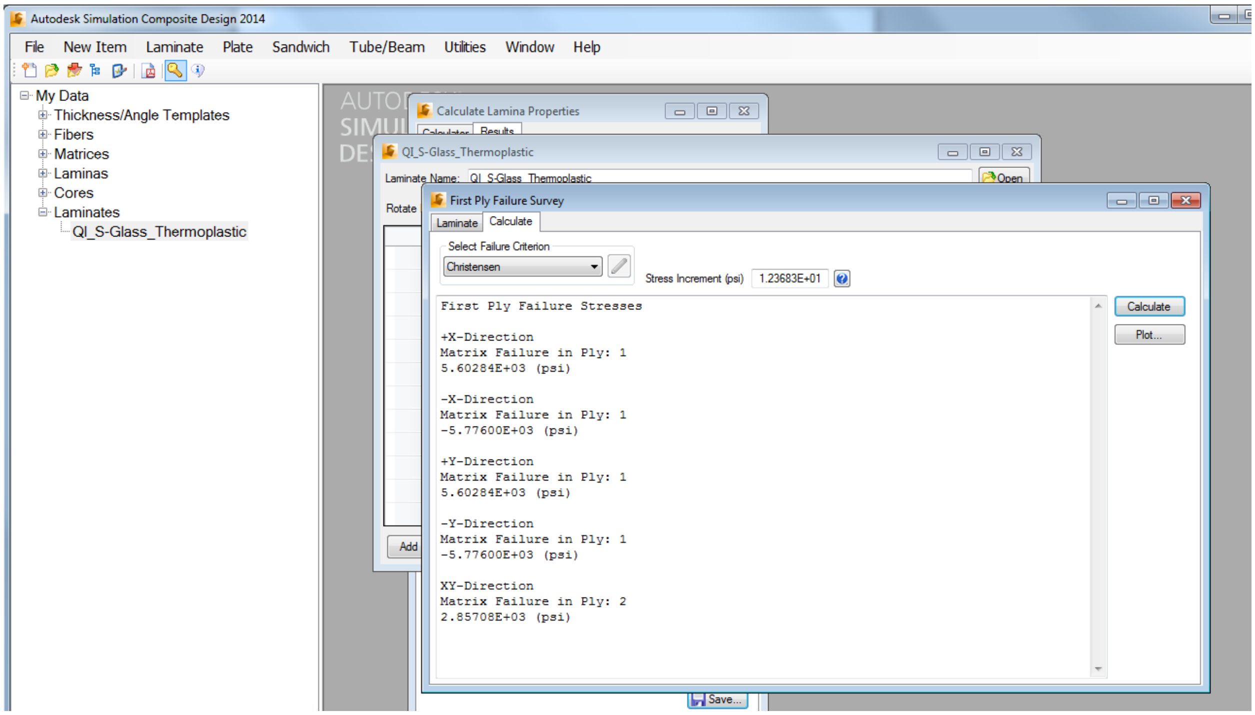
Second, our 24 ply quasi-isotropic laminate can be created using this newly created lamina material.



Next, we can determine the stiffness and other engineering properties of our new composite laminate using the **Laminate Properties** tool under the Laminate menu.

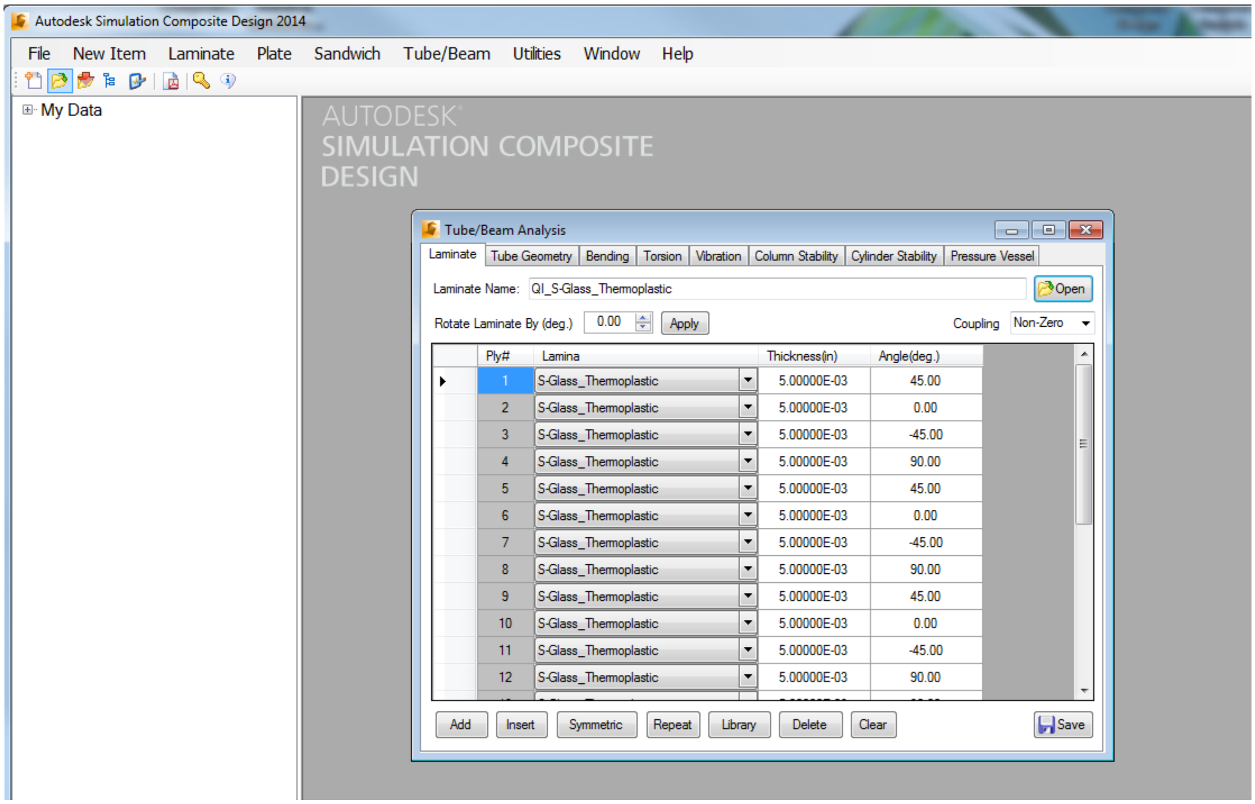


Finally, we can determine the strengths of our composite laminate using the **First Ply Failure Survey** tool under the Laminate menu. For this example, we will use the Christensen failure criterion to determine which ply in our composite laminate will fail first when loaded in the X-direction in tension and compression, the Y-direction in tension and compression, and in shear. The Christensen failure criterion is able to tell us that these first-ply-failure are matrix failures for all 5 loading directions. If we want to determine the ultimate strength of our composite laminate, we could use the *Progressive Failure* tab in the **Stress-Strain and Strength Analysis** tool under the Laminate menu.

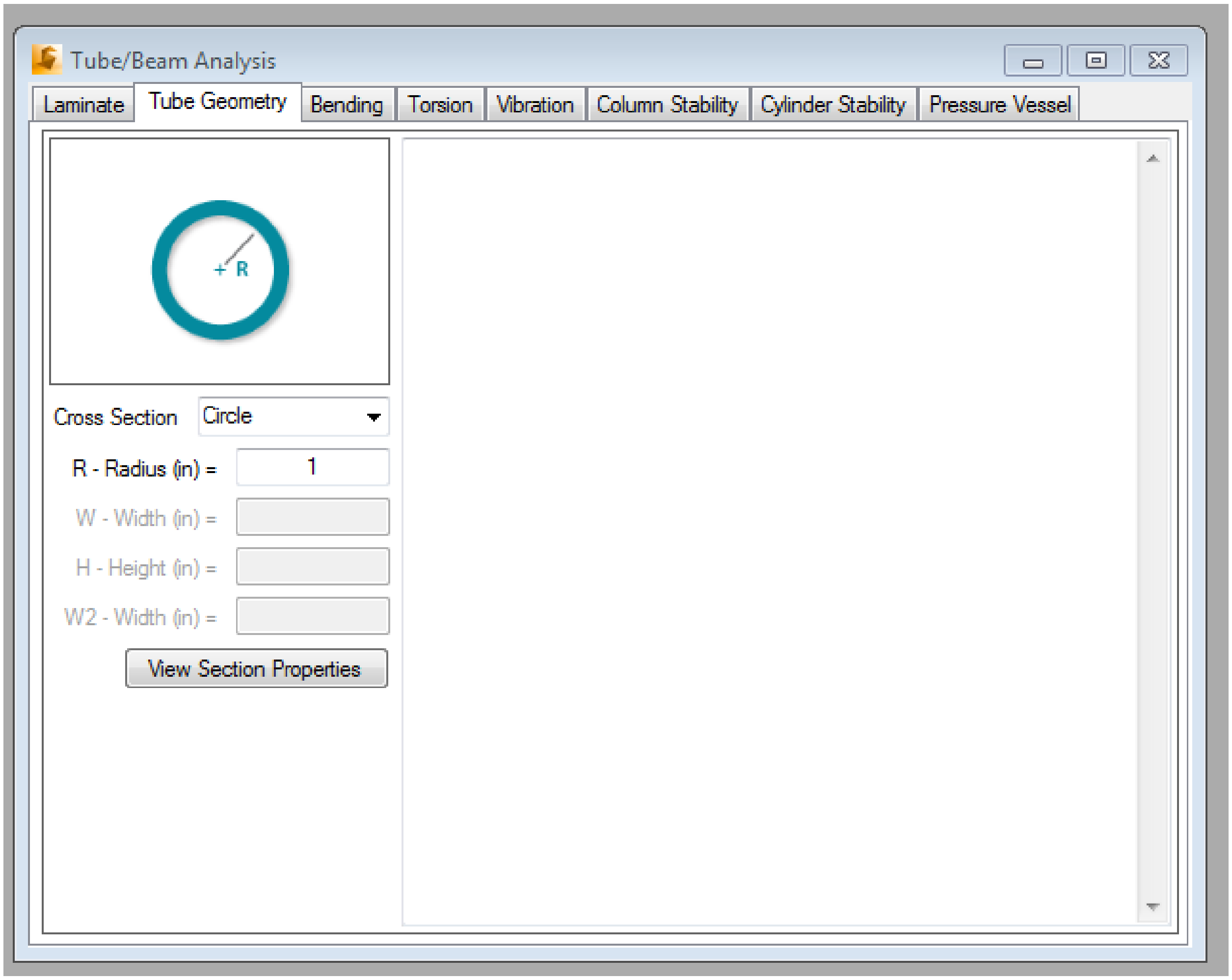


Example 2 – Determine the Deflection and Internal Stresses of a Shovel Handle Due to a Prying Load

In this example, we will use the laminate created in Example 1 to determine the deflection and internal stresses of a shovel handle that is being used to pry out a system of tree roots. First, we can open the Tube/Beam Analysis tool under the Tube/Beam menu, and import our laminate.

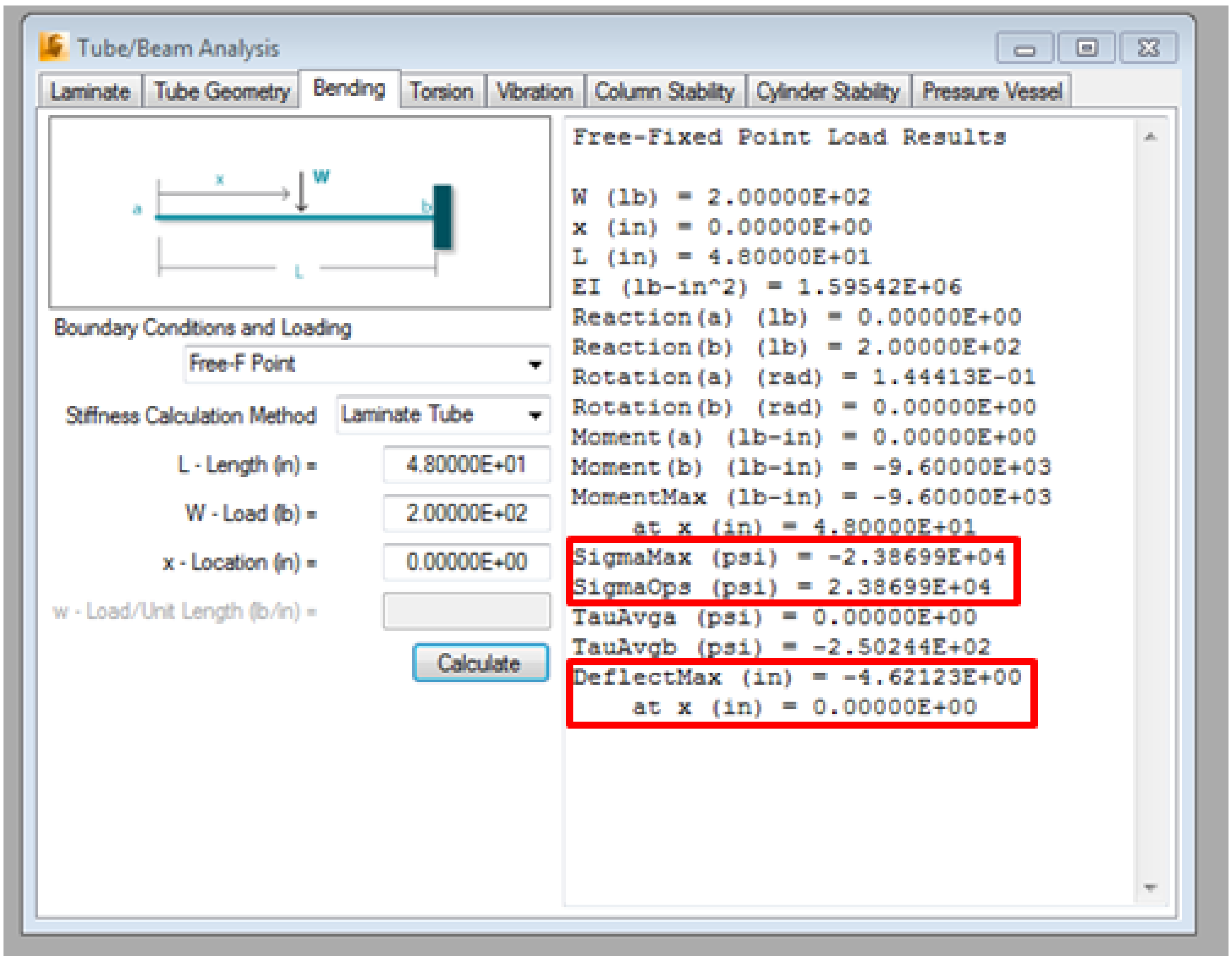


Second, we can define our shovel handle geometry under the *Tube Geometry* tab as a circle cross section with a radius of 1 in.



Finally, under the *Bending* tab, we can define the prying load and boundary conditions. We will define the Boundary Conditions and Loading as a Free-F Point (Free-Fixed beam boundary condition with a point load) to represent a 200 lb. man using his entire body weight to pry out the tree roots with the shovel head fixed in the ground.

The length of our shovel handle is 48 in., with a load of 200 lb. applied at the tip of the shovel handle. By clicking on the Calculate button, we can see that the maximum deflection of the handle tip is 4.6 in. and the maximum stresses in the tube are 23.9 ksi in compression and tension (depending on the side of the tube in bending) at the shovel head.



Autodesk® Simulation Composite Analysis

What is it?

Autodesk Simulation Composite Analysis is a plugin to conventional finite element analysis (FEA) codes such as Abaqus and ANSYS. The plugin aids users in performing progressive failure analyses of composite structures by providing material properties of composite lamina, providing material characterizations for modeling delamination using cohesive elements, and providing unsurpassed convergence technology to aid FEA in simulating composite failure after first ply failure (FPF) predictions.

What does it do?

Autodesk Simulation Composite Analysis bolts on to conventional FEA codes to aid with progressive failure analysis of composite materials. Supported FEA codes currently include

- Abaqus
- ANSYS

Autodesk Simulation Composite Analysis comes predefined with many unidirectional and woven composite laminae. If a user wishes to characterize their own material systems, they can use Material Manager to characterize:

- Unidirectional lamina
- Plain woven lamina
- 4 harness satin weaves
- 5 harness satin weaves
- 8 harness satin weaves

All characterized materials can be shared and used in the supported FEA programs to define composite material definitions using Autodesk Simulation Composite Analysis Plugins. Supported options for progressive failure analyses include

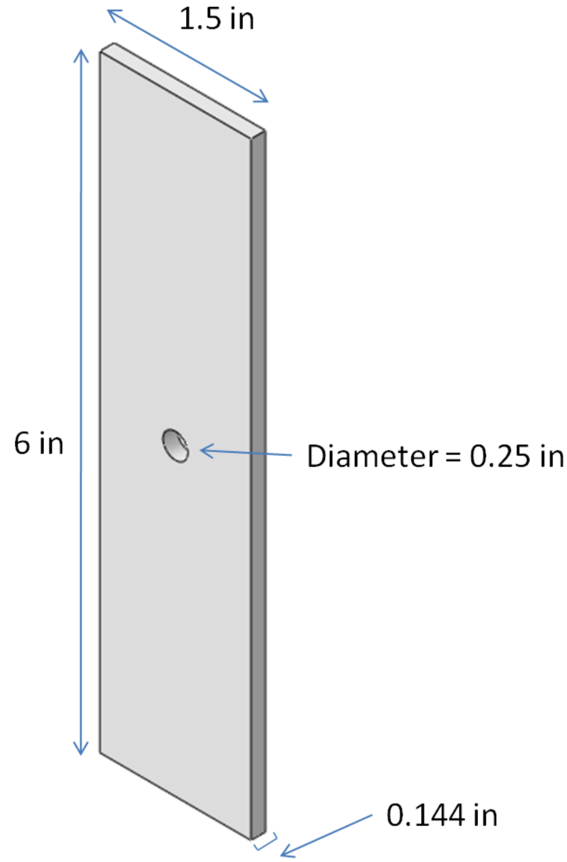
- Availability of multiple failure criteria
- Availability to perform progressive fatigue analysis
- Instantaneous degradation of the composite after a failure event
    - Degradation can be separated and performed at the constituent level
- Mesh independent degradation after failure events
- Cohesive property definitions for delamination analysis

Post-processing within the FEA environment is also efficient by the use of additional state variables which can be overlaid on contour plots. The additional information allows users to determine which types of failure are occurring and view failure indices of constituents.

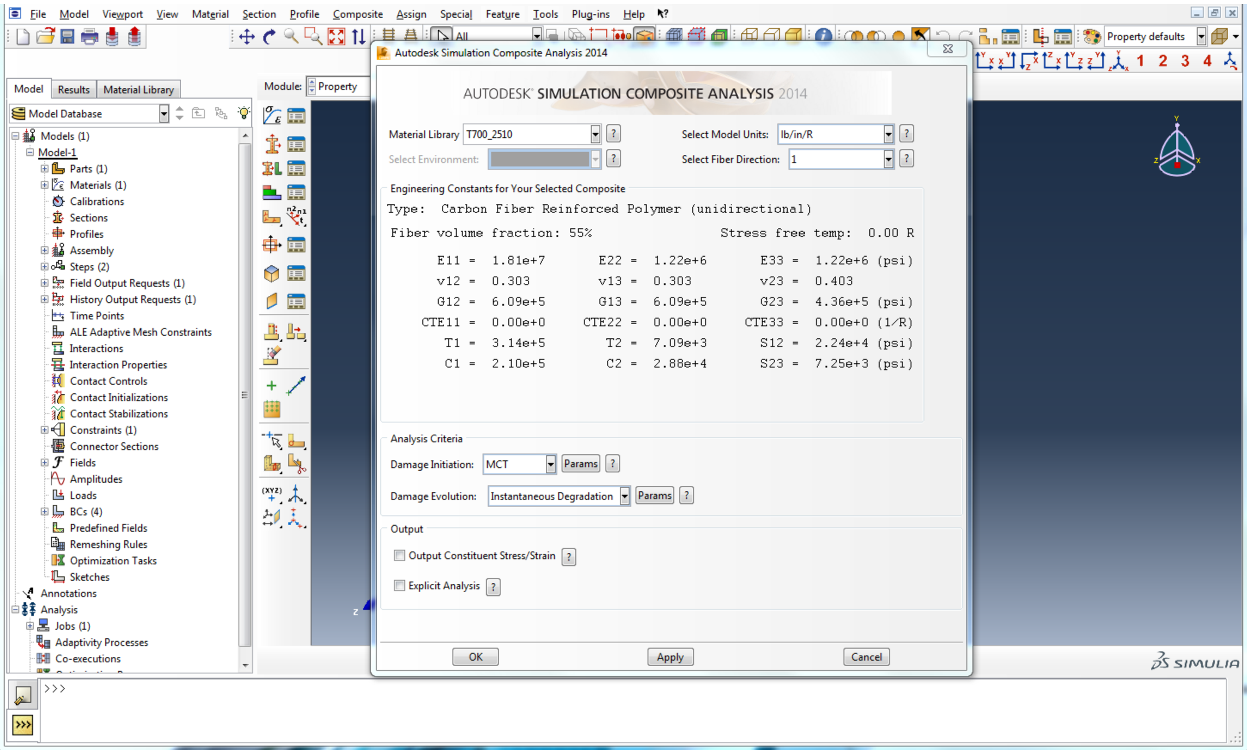
The ability to couple intra and inter-laminar failure and converge on an entire load history, past initial failure, is uncommon in composites simulation. The technology within Autodesk

Simulation Composite Analysis allows for unsurpassed convergence and confidence for progressive failure analyses.

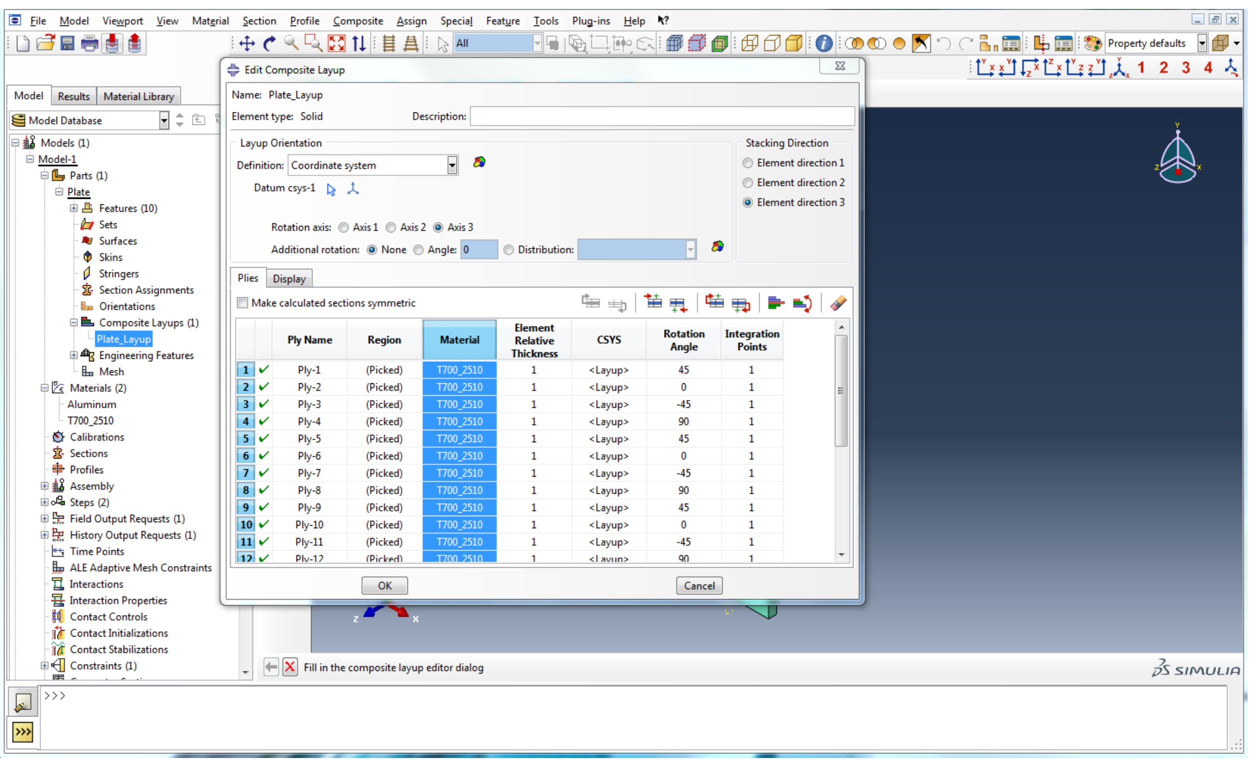
Example – Determine Open-Hole-Tension Strength of Composite Coupon
For this example, we will determine the strength of an open-hole-tension (OHT) composite coupon. This coupon is a $[45/0/-45/90]_{3S}$ laminate made from layers of unidirectional T700/2510 carbon-epoxy plies. We will run a progressive failure simulation using the Autodesk Simulation Composite Analysis plug-in within the FEA software package Abaqus.



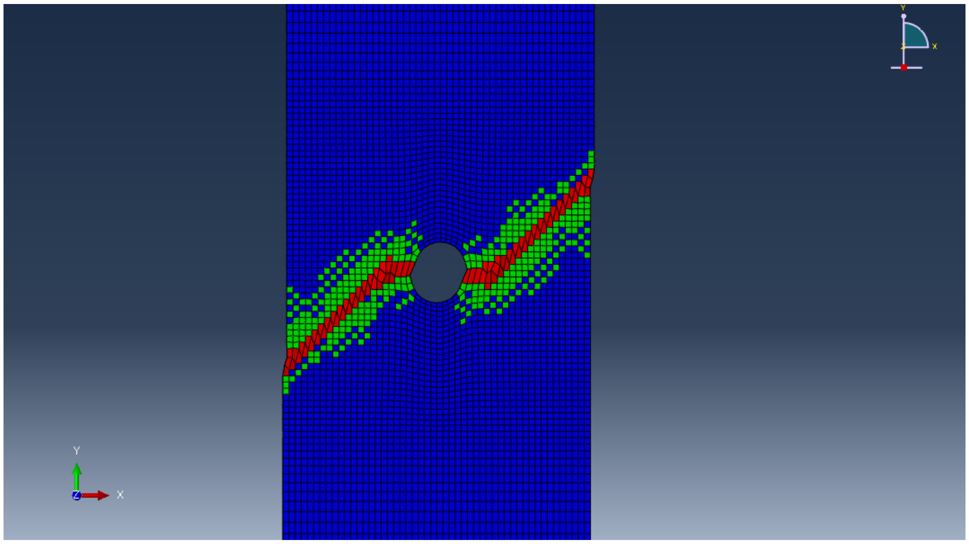
After the coupon geometry, mesh, loads, and boundary conditions are created for this model, the material model for T700/2510 is accessed from the composite material library that comes with Autodesk Simulation Composite Analysis. To create a composite material, the plug-in is accessed from the Abaqus/CAE Plug-ins menu. The creation of a composite ply material is the only change to a standard FEA analysis required to get the progressive failure information for the OHT coupon.



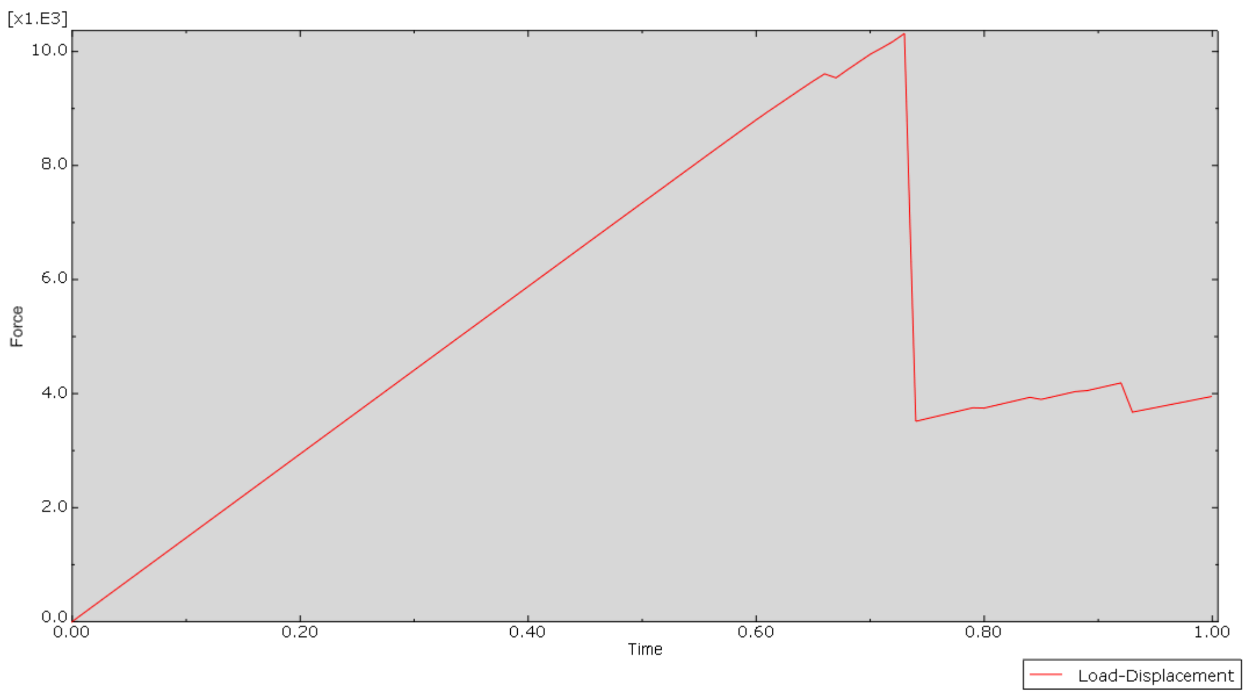
The next step is to assign this newly created material to a composite layup.



Finally we can run the analysis and view the results. The contour plot shows which areas of the coupon have failed: blue represents a pristine material, green represents regions of the coupon that have matrix failure, and red represents regions of the material that have fiber failure.



To determine the strength of the OHT coupon, we can generate a load-displacement plot that shows the peak load the coupon is able to withstand before there is enough accumulation of damage to cause catastrophic failure.



Summary

In this discussion we have introduced two new products for the design and analysis of composite structures. We have provided use cases and examples for each product and given the context and scenarios in which each product may be useful for engineers and analysts who require more knowledge early in the design process.