# WoundSim Abaqus

The **WoundSim Abaqus** extension is an Abaqus/CAE plugin that allows a user to read the .xml output file from the **WoundSim GUI**, then build a complete, run-ready Abaqus finite element model. Alternatively, a user can build an Abaqus model, run it, and post-process it entirely within the **WoundSim Abaqus** extension. Some of the tools include advanced material properties allowing for axisymmetric continuum modeling of the COPV, and the ability to copy and modify a COPV design and quickly rebuild, run, and display a comparison of output plots such as fiber strains between the designs.
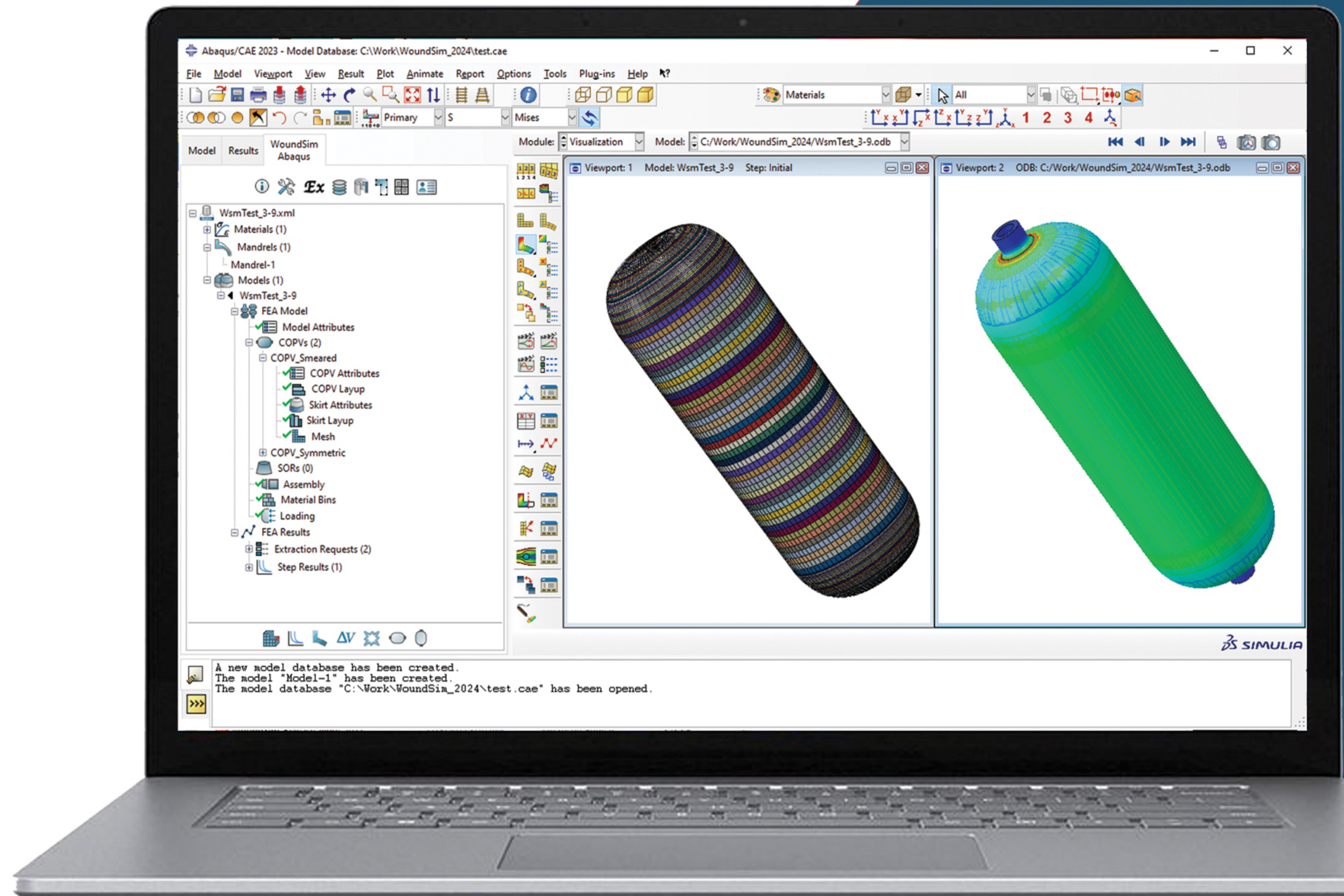
- » Axisymmetric shells and continuum element support using smeared properties representing the +/- fiber directions

- » 3D shells using smeared properties or modeling each layer as 4-ply symmetric layup which allows progressive failure simulation

- » 3D standard bricks, continuum-shell bricks, layered bricks, and cylindrical brick element.

- » Layered brick and continuum shell can be based on smeared or 4-ply symmetric layer

- » Micro-mechanics modeling allowing lamina property creation from fiber and matrix properties

- » Automated path plots from output requests

- » Multiple COPVs in a single model

- » Output path plots can be stored for multiple designs for easy design comparisons

- » Applications include cryogenic, autofrettage, drop test, impact, burst, and more

For More Information:

Please see our website at qustomapps.com, or contact Maria Shubert at

✉ maria.shubert@qustomapps.com

WoundSIM

Design and Simulation of
Composite
Pressure Vessels

WoundSim is a worldwide leading software tool used to design and simulate composite overwrapped pressure vessels (COPVs). The WoundSim GUI is used to instantly view the composite layup from a table of composite layer definitions. Layer thickness buildup is automatically calculated, along with the continuously varying wind angles. Material properties are computed and assigned throughout the COPV.
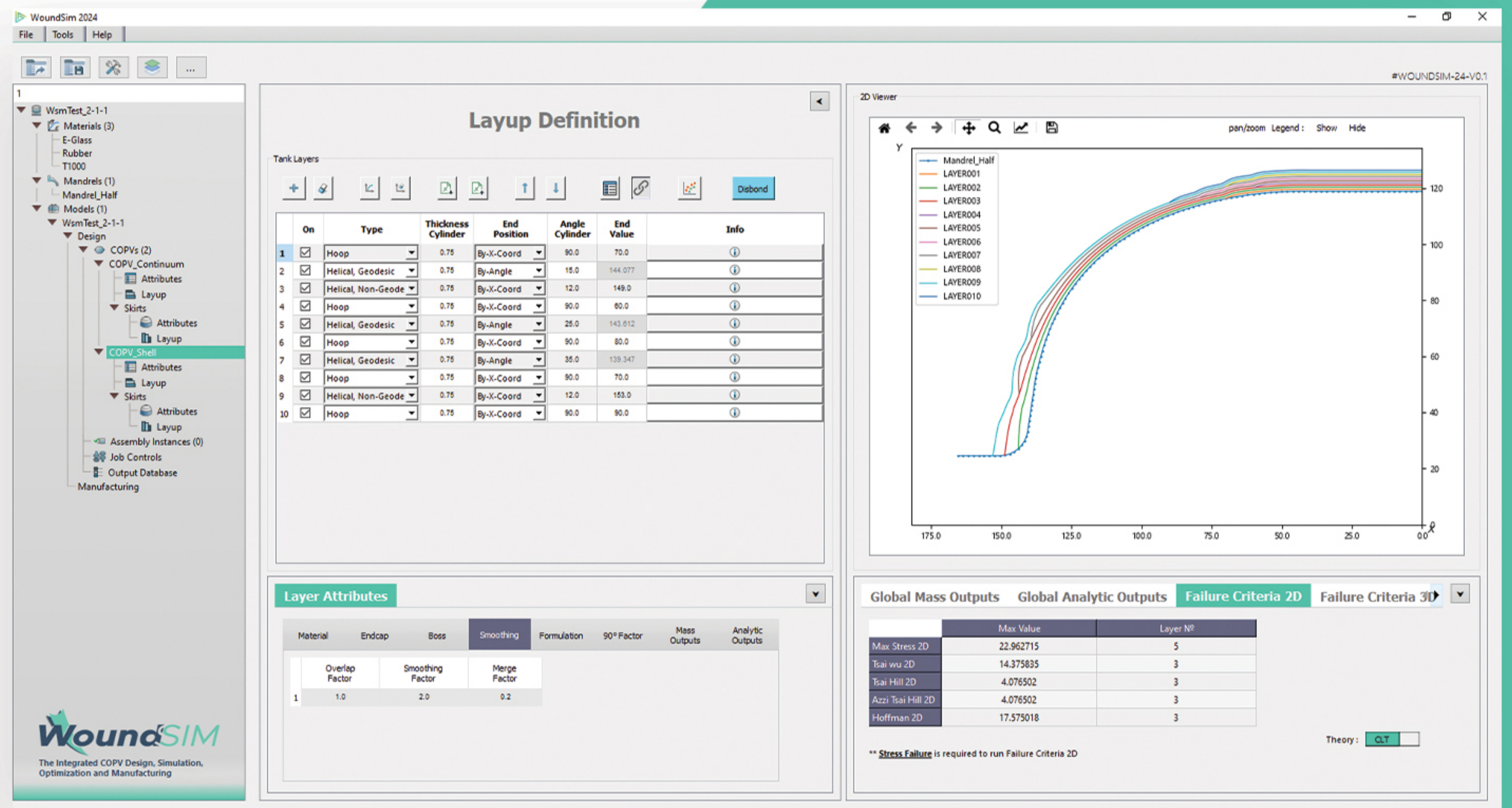
WoundSim GUI

The standalone **WoundSim GUI** allows a user to define the layup of a COPV and export that layup into the WoundSim Abaqus extension to build run-ready finite element models.

» Comprehensive and well-chosen design parameters allowing a quick variation of the layers shape.

» Layer types include helicals with and without friction, hoops, and doilies

» Fully automated FEA model generation and post processing, allowing creation of simulations with minimum FE knowledge.

» Analytic Pre-Design capabilities, allowing initial assessment of the design

» Failure measure pre-design calculation can be performed for thin or thick shell theory

» Automated path plotting of a large selection of output quantities

» Once model has been built in WoundSim Abaqus, any changes to the model can be rebuilt, run, and output results generated with just a couple mouse clicks

For More Information:

Please see our website at qustomapps.com, or contact Maria Shubert at

✉ maria.shubert@qustomapps.com