

SIEMENS

Ingenuity for life

Multimech for Simcenter 3D

Simplify converting a Simcenter Nastran model into a multiscale model

Benefits

- Simplify converting a Simcenter Nastran model into a multiscale model
- Deliver accurate modeling of progressive damage, including crack initiation and growth
- Provide generic framework for any multiphase material
- Automatically generate microstructures
- Reverse engineer constituent properties
- Account for manufacturing-induced variation

Summary

Multimech for Simcenter™ 3D software allows users to easily convert existing models into True Multiscale models.

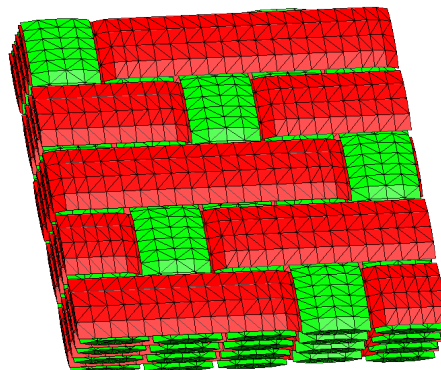
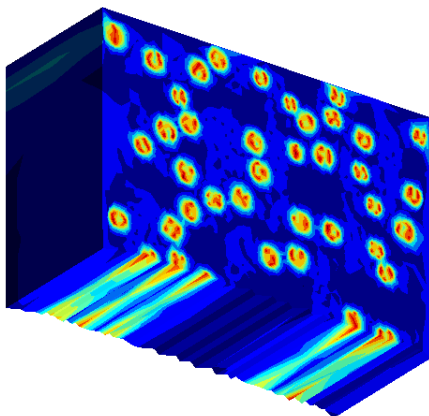
Using this application, engineers can make design choices based on high-fidelity multiscale simulation that pinpoints the origin of failure at the microstructure.

- See how homogenized properties change in space and time
- Zoom into microscale to assess damage and relay that information to the macroscale
- Achieve more accurate simulation results due to representing behavior at the microscale

How it works

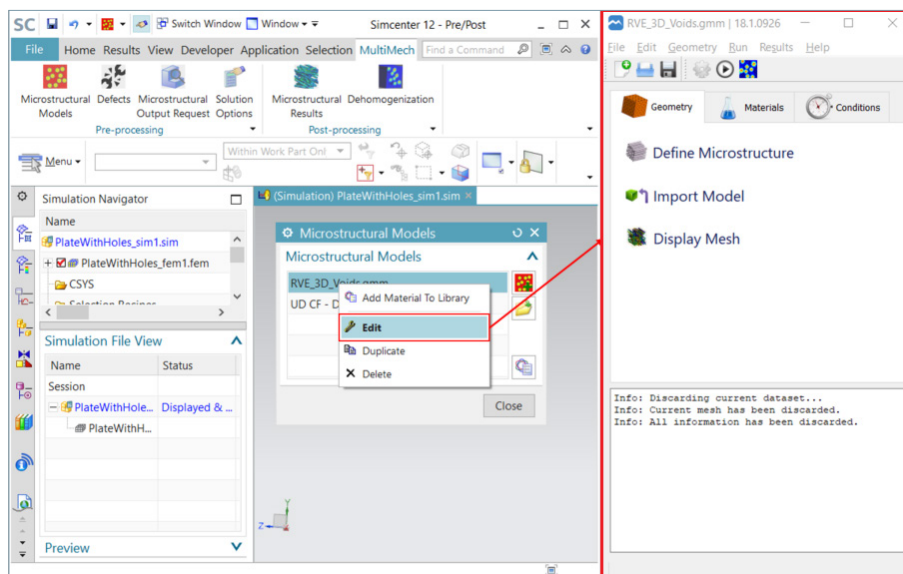
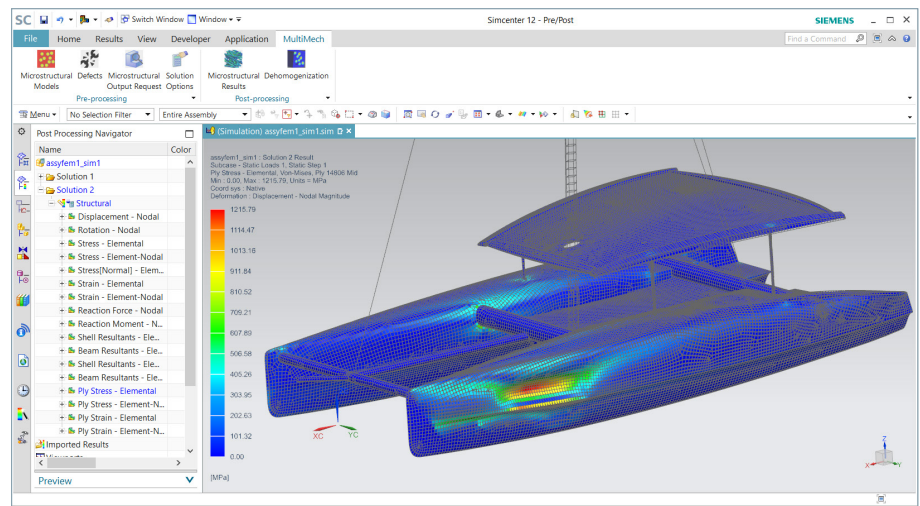
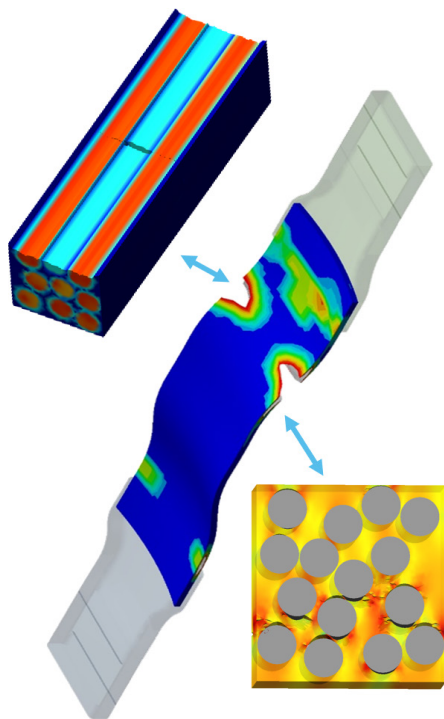
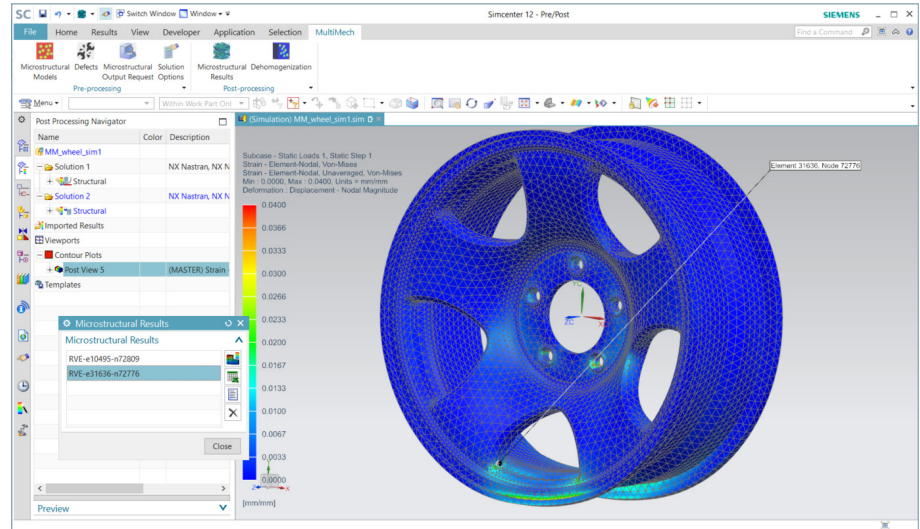
Multimech finite element (FE) software from Siemens Digital Industries Software is integrated with the Simcenter 3D software user interface, making it simple to convert an existing Simcenter Nastran® software model into a full True Multiscale model. After the Multimech for Simcenter 3D application is installed, a new ribbon tab named Multimech is added to the pre/post application.

1. The microstructural model functionality allows users to create or import microstructures and create Simcenter materials that can be used in Simcenter Nastran simulations.
2. Creating a new microstructure or editing an existing one launches the Multimech user interface, allowing the user to define or edit the microstructural model.
3. Once microstructural materials are created and assigned to parts, the full multiscale simulation will be run to solve the model, automatically coupling Simcenter Nastran with the Multimech solver in a concurrent fashion. In addition to the traditional post-processing capabilities available in Simcenter 3D, the Multimech extension enables users to postprocess microstructural results as well. For example, users can see how homogenized properties change in space and time and can perform full postprocessing of microstructural results at any location of the part model to rigorously investigate when/where/why damage originates at the microscale.



The Multimech for Simcenter 3D tool uses the extremely detailed and accurate concurrent multiscale approach. Using this approach, an engineer can have a global scale model with a part that is subjected to different loading and boundary conditions. The stresses and strains within this part are transferred to a specified microscale finite element model, which represents each integration point in the global scale model.

Once the microscale model is solved that information is passed to the global scale, enabling much more realistic simulation results by capturing history dependence of material laws and stress redistribution due to damage evolution.



Siemens Digital Industries Software
[siemens.com/software](https://www.siemens.com/software)

Americas +1 314 264 8499
 Europe +44 (0) 1276 413200
 Asia-Pacific +852 2230 3333