

MultiMech

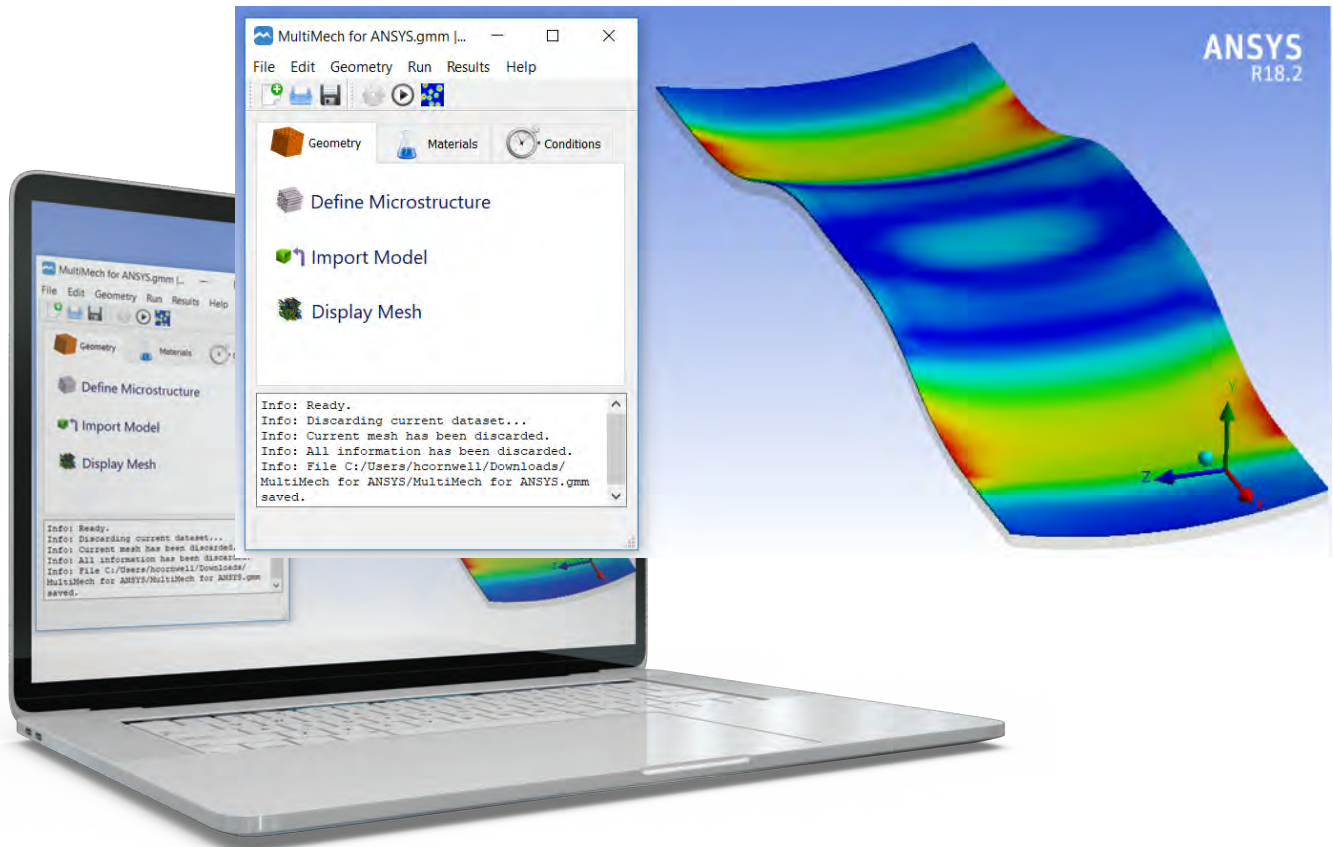
FOR

ANSYS®

LANIKA SOLUTIONS PRIVATE LIMITED

TF-04, Gold Signature, No. 95, Mosque Road, Frazer Town, Bangalore - 560 005, INDIA

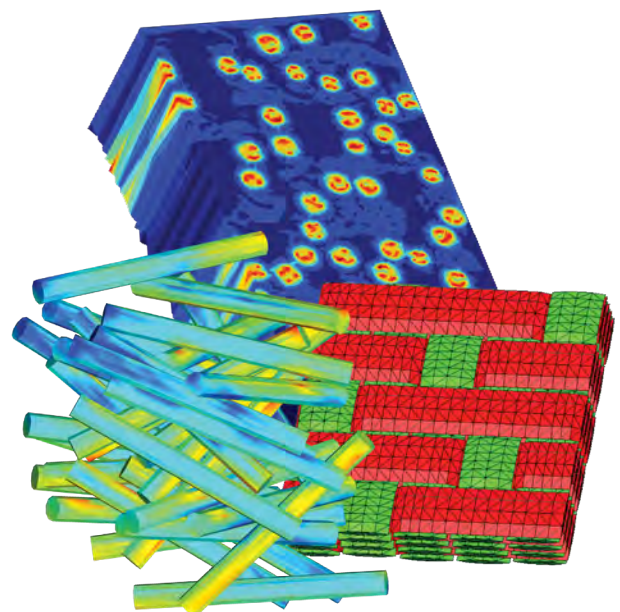
Phone: +91 – 80 – 2548 4844 Fax: +91 – 80 – 2548 4846 Email: info@lanikasolutions.com www.lanikasolutions.com

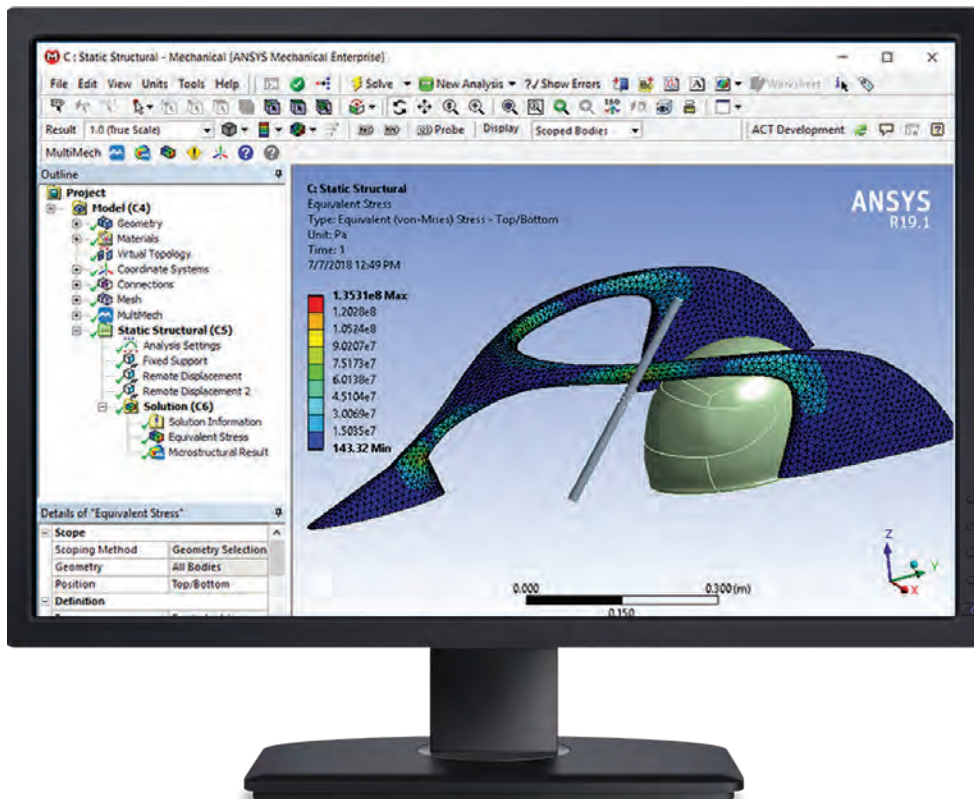


MULTIMECH FOR ANSYS ALLOWS WORKBENCH USERS TO EASILY CONVERT EXISTING MODELS INTO TRUE MULTISCALE MODELS.

Using this plugin, engineers are enabled to make design choices based on the high-fidelity multiscale simulation that pinpoints the origin of failure at the microstructure.

- Observe microstructure properties
- Zoom into microscale to assess damage and relay to the macroscale
- Achieve more accurate simulation results due to true representation of behavior at microscale





GENERAL FEATURES

Integrated within
ANSYS native
user interface

Automatic
generation of
microstructures

Accurate
characterization
of progressive
damage

Automatic
crack initiation
and growth

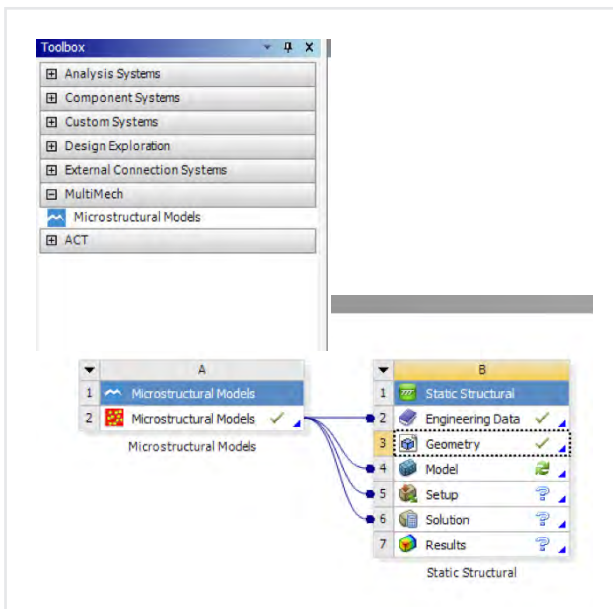
Structural
and material
optimization

Material
calibration via
optimization

Accurately model
any multiphase
material

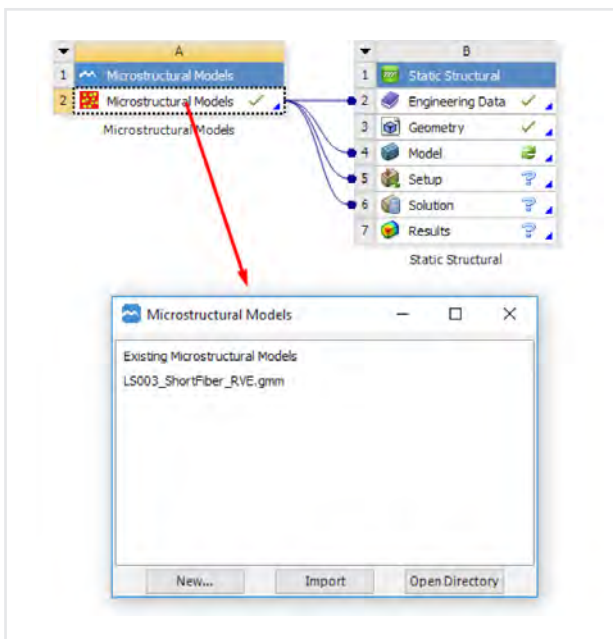
HOW IT WORKS

MultiMech is integrated within the ANSYS native user interface, making it very simple to convert an existing ANSYS model into a full TRUE multiscale model. After the MultiMech extension is installed, it is possible to create or import microstructures to be imported as materials in the model. These materials become available to be assigned to parts within ANSYS Mechanical, after which a multiscale simulation is automatically executed when the model is solved.



1

After the extension is installed, a MultiMech object will be present in ANSYS Workbench, as seen in the image to the left. Connecting it with the Engineering Data object of a model will make the microstructures selected available for use as materials in ANSYS Mechanical.

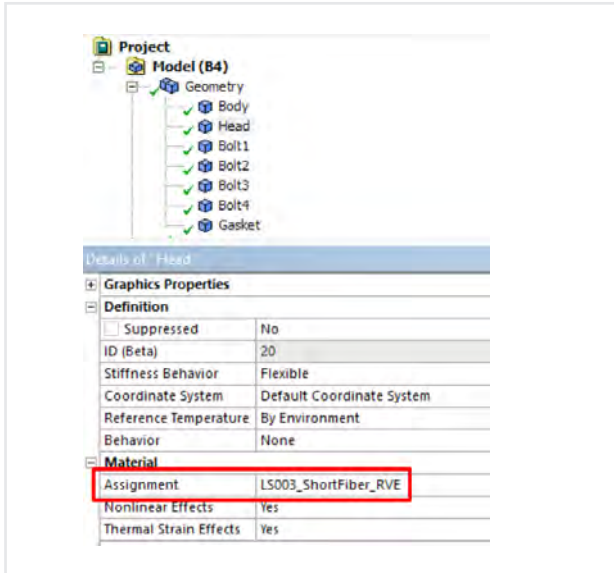


2

To select a microstructure, open the Microstructural Models object, then hit **New** to open MultiMech GUI and define a new microstructure. Alternatively, an existing RVE can be imported.

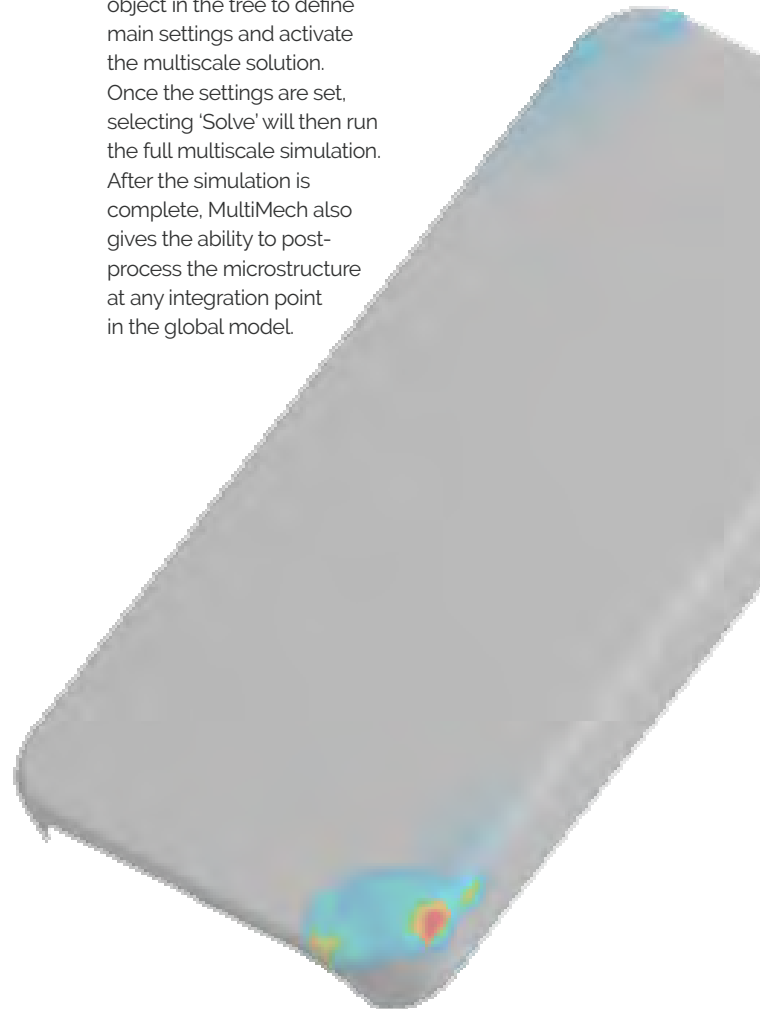
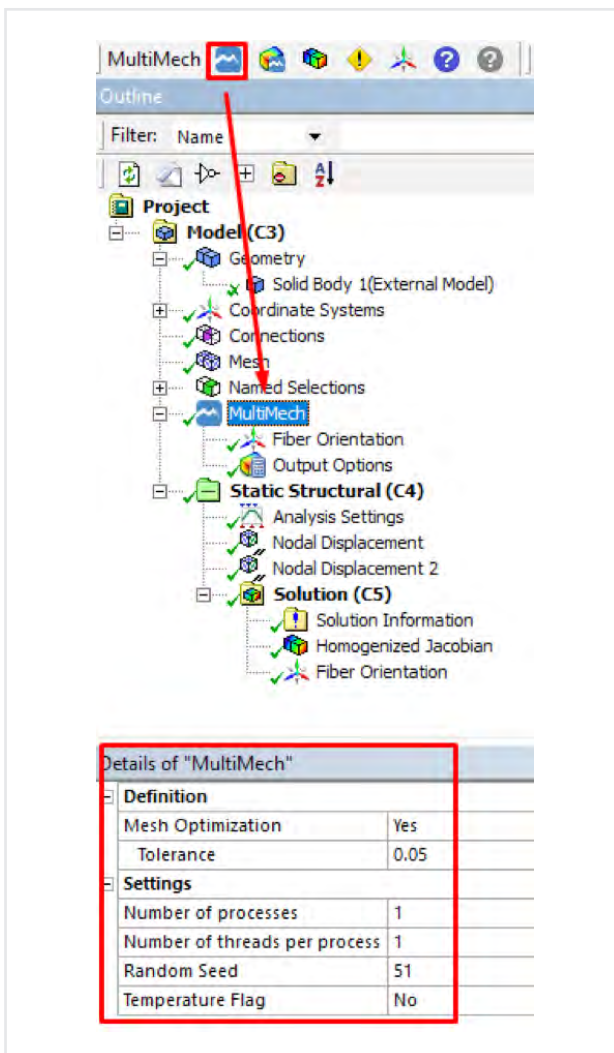
3

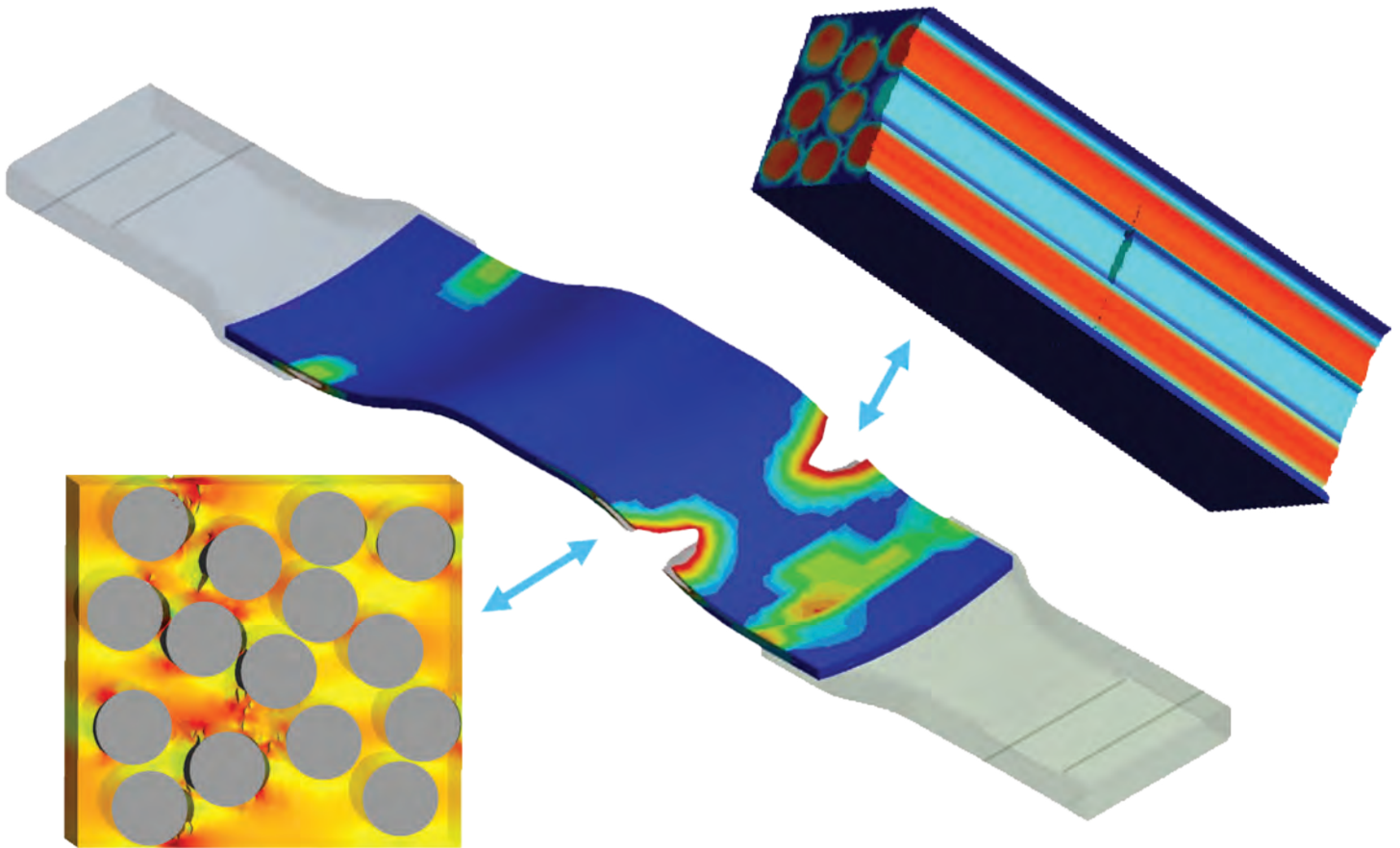
When ANSYS Mechanical is opened, all the user needs to do is to assign the microstructures as materials to specific parts...



4

... then create a MultiMech object in the tree to define main settings and activate the multiscale solution. Once the settings are set, selecting 'Solve' will then run the full multiscale simulation. After the simulation is complete, MultiMech also gives the ability to post-process the microstructure at any integration point in the global model.





The MultiMech for ANSYS tool utilizes 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 within 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.

ABOUT MULTIMECHANICS

MultiMechanics was founded in 2010. One of its first projects was helping the US Army Research Laboratory explore impact applications, and the company has since evolved to specialize in helping material manufacturers and their clients better understand how composites will behave under extreme conditions. Our software helps companies run composite simulations at an unprecedented level of speed and accuracy.

MultiMechanics is based in Omaha, Nebraska. The company completed its first round of seed funding in 2017 and received a co-investment of \$1.9 million from Solvay Ventures, Anzu Partners, and Invest Nebraska. Since then, the MultiMechanics team has more than doubled in size.



hello
tomorrow

TOP 7
STARTUPS IN NEW MATERIALS

CIO
Review

20 MOST PROMISING
Modeling/Simulation
SOLUTION PROVIDERS - 2016



xC Startup Award
TOP 10



Editor's Pick
of the Week



INNOVATOR OF THE YEAR
FINALIST





MultiMechanics

**Virtual Testing Software For
The World's Most Advanced Materials**

MultiMechanics.com

© 2018 MultiMechanics, Inc

LANIKA SOLUTIONS PRIVATE LIMITED

TF-04, Gold Signature, No. 95, Mosque Road, Frazer Town, Bangalore - 560 005, INDIA

Phone: +91 – 80 – 2548 4844 Fax: +91 – 80 – 2548 4846 Email: info@lanikasolutions.com www.lanikasolutions.com