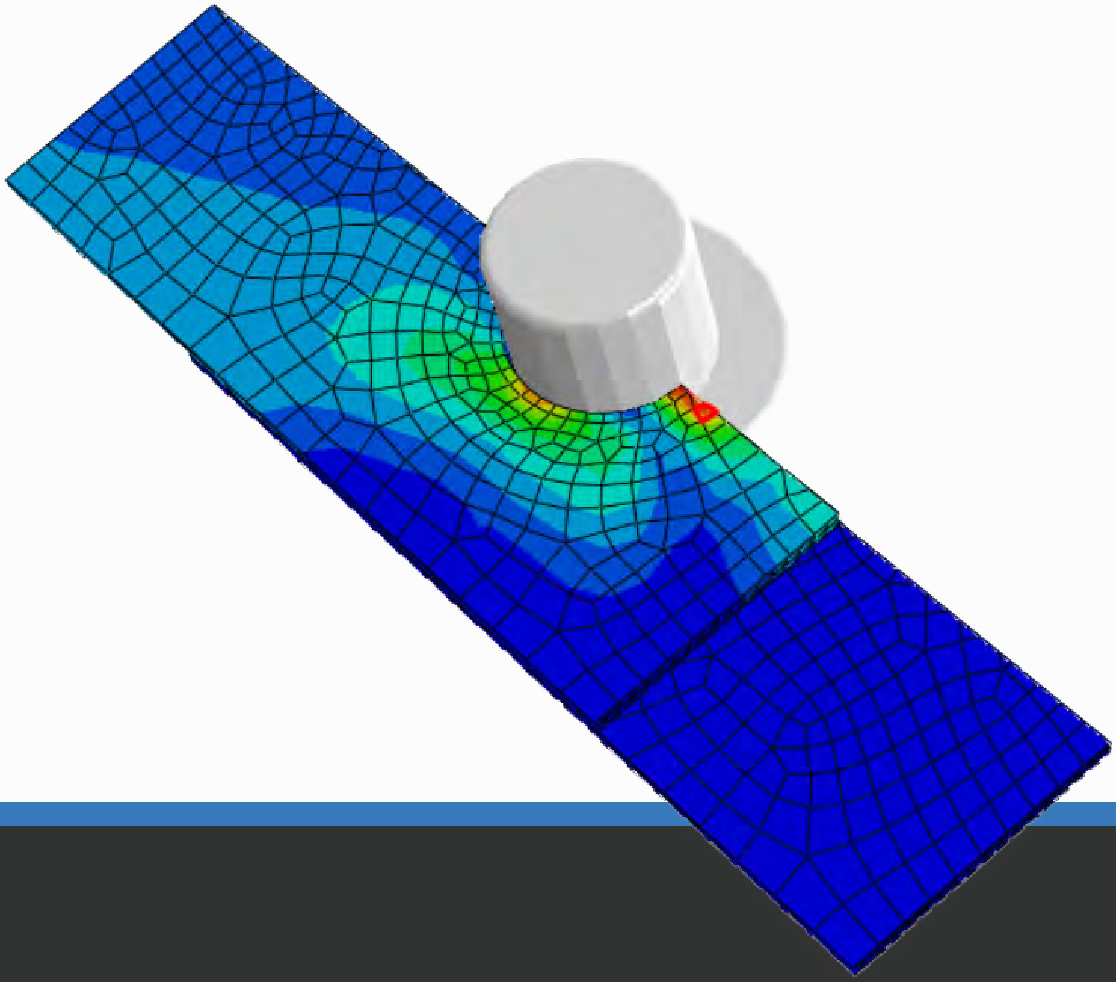
LANIKA SOLUTIONS



MultiMech

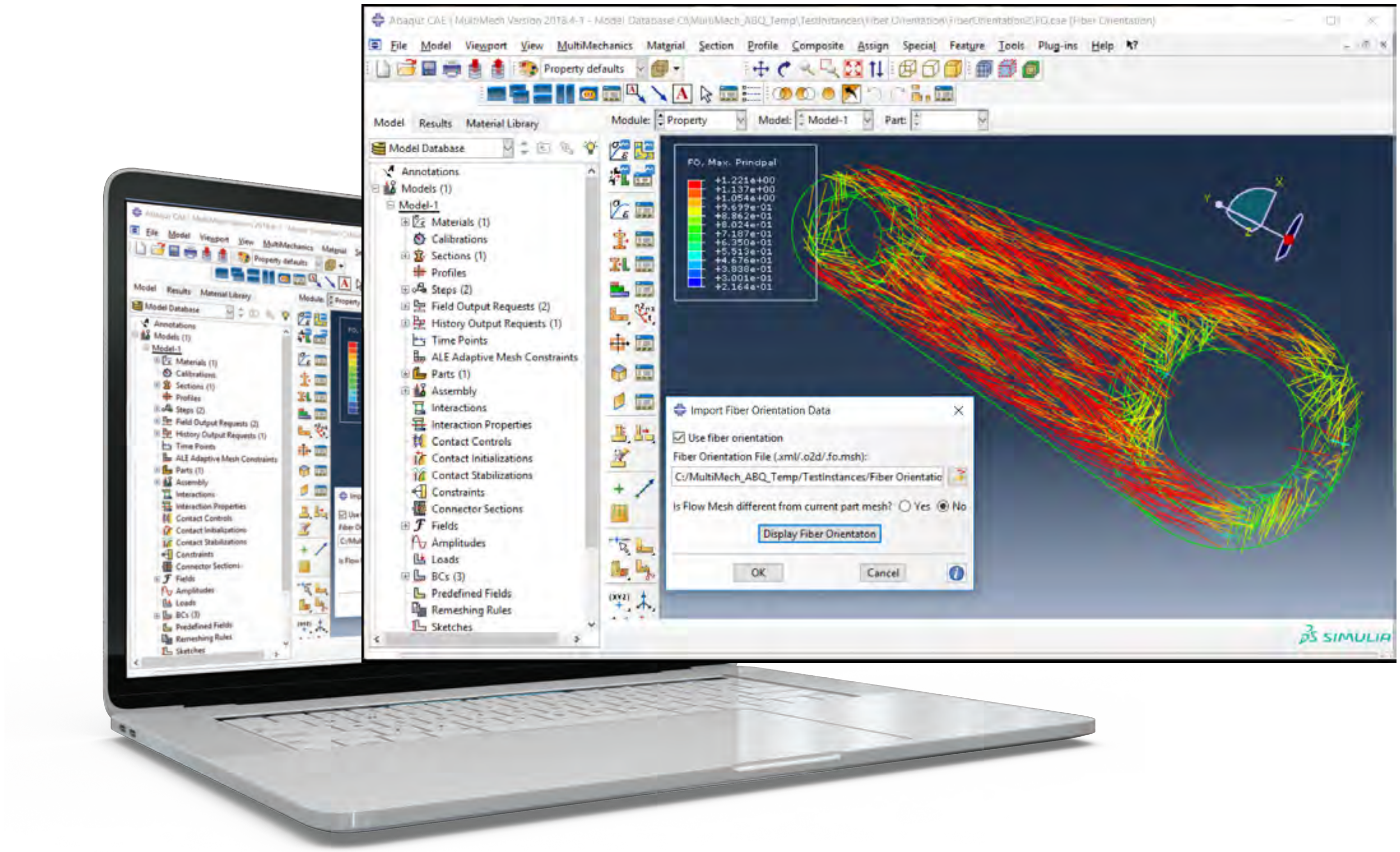
FOR

ABAQUS

LANIKA SOLUTIONS PRIVATE LIMITED

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

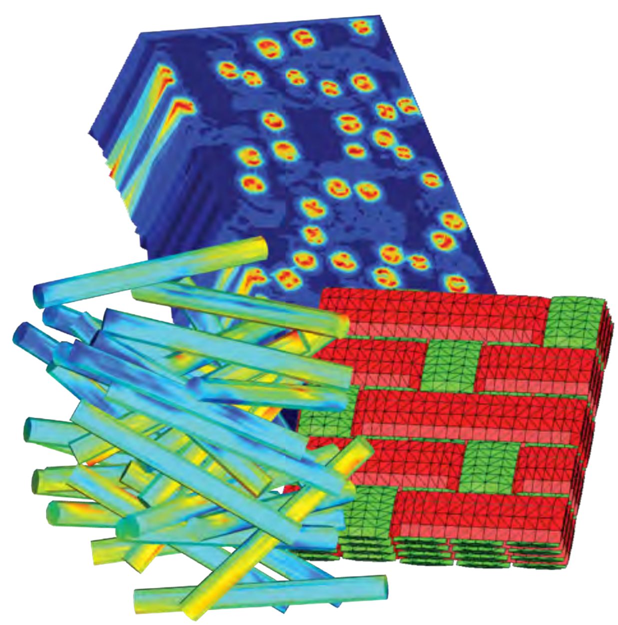
Phone: +91 – 80 – 2548 4844 Fax: +91 – 80 – 2548 4846 Email: info@lanikasolutions.com www.lanikasolutions.com

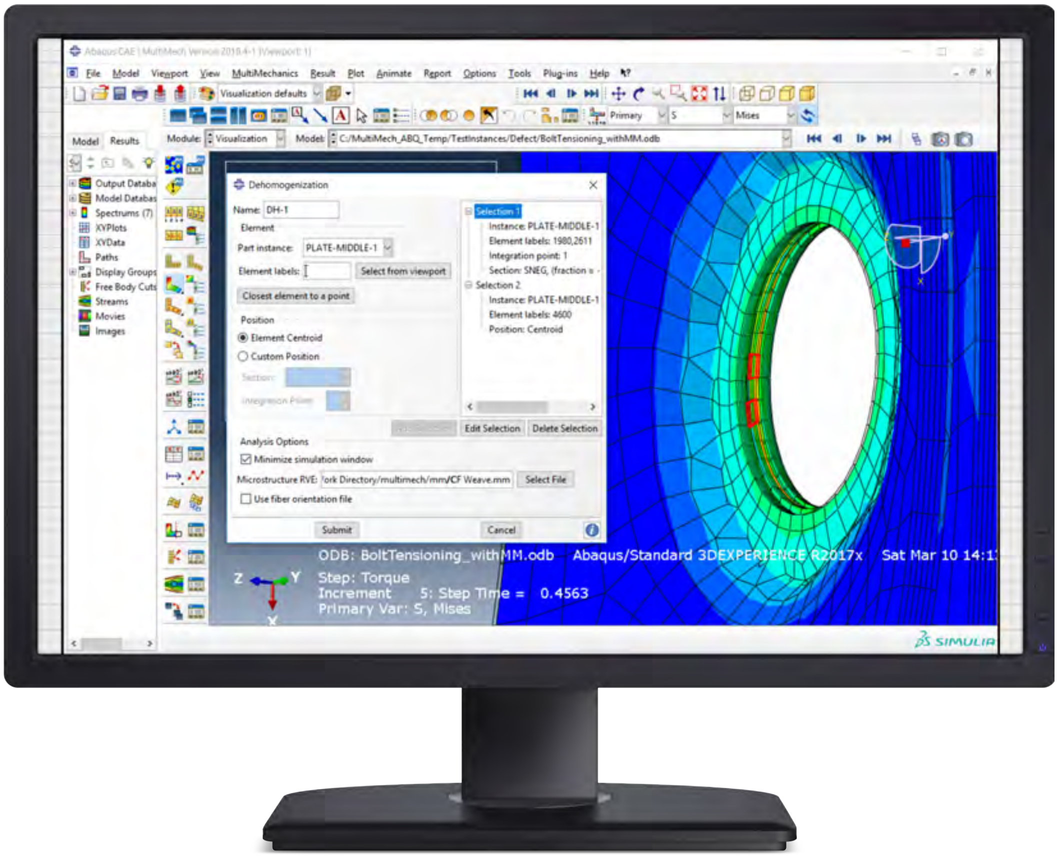


MULTIMECH FOR ABAQUS ALLOWS USERS TO EASILY CONVERT EXISTING MODELS INTO TRUE MULTISCALE MODELS.

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 the microscale



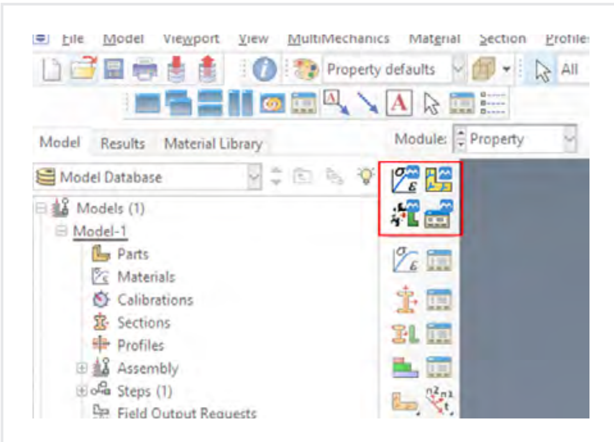


GENERAL FEATURES

- Integrated within Abaqus 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

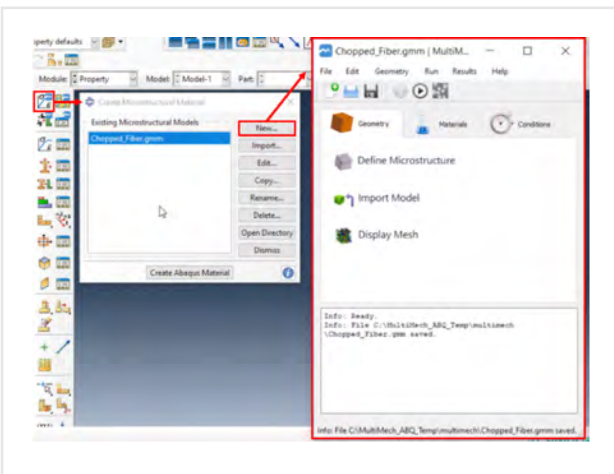
HOW IT WORKS

MultiMech is integrated within the Abaqus native user interface, making it very simple to convert an existing Abaqus model into a full TRUE multiscale model. After the MultiMech for Abaqus extension is installed, multiple MultiMech options will be available in the different modules within Abaqus. This gives the user the ability to create a microstructure model and define its constituent material properties, along with many other capabilities, all within Abaqus.



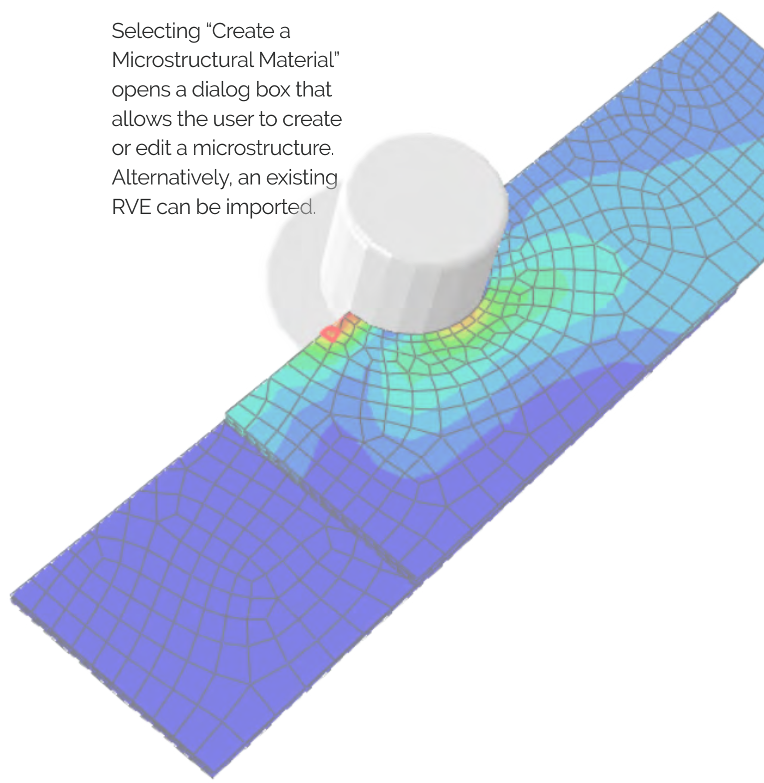
1

Once the extension is installed, new icons with the MultiMech logo will appear in the "Property" module, as seen in the image above. These icons allow for the different capabilities of MultiMech to be used within Abaqus, such as creating microstructural models, importing material orientations, or inserting defects.

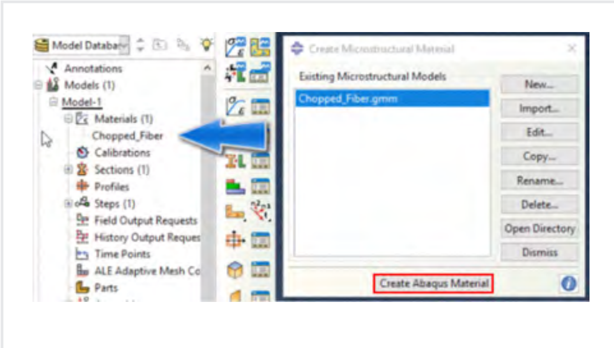


2

Selecting "Create a Microstructural Material" opens a dialog box that allows the user to create or edit a microstructure. Alternatively, an existing RVE can be imported.

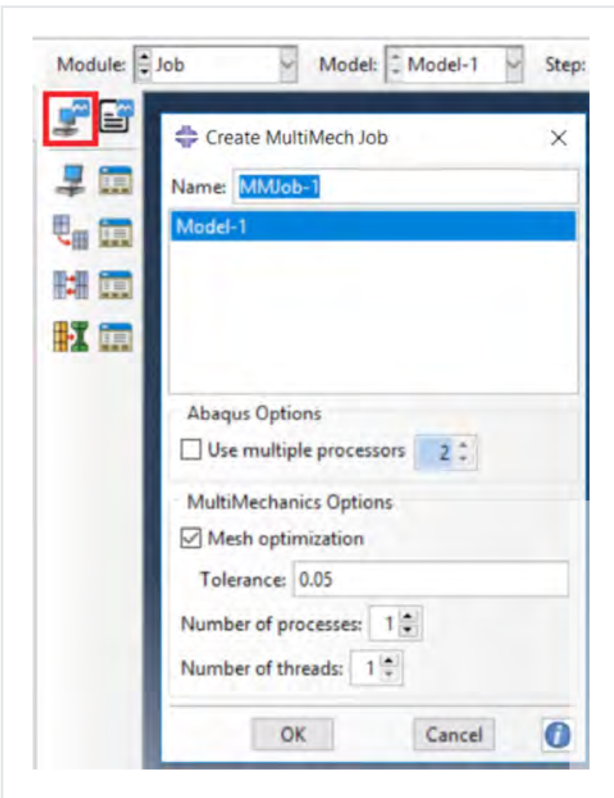


MultiMechanics



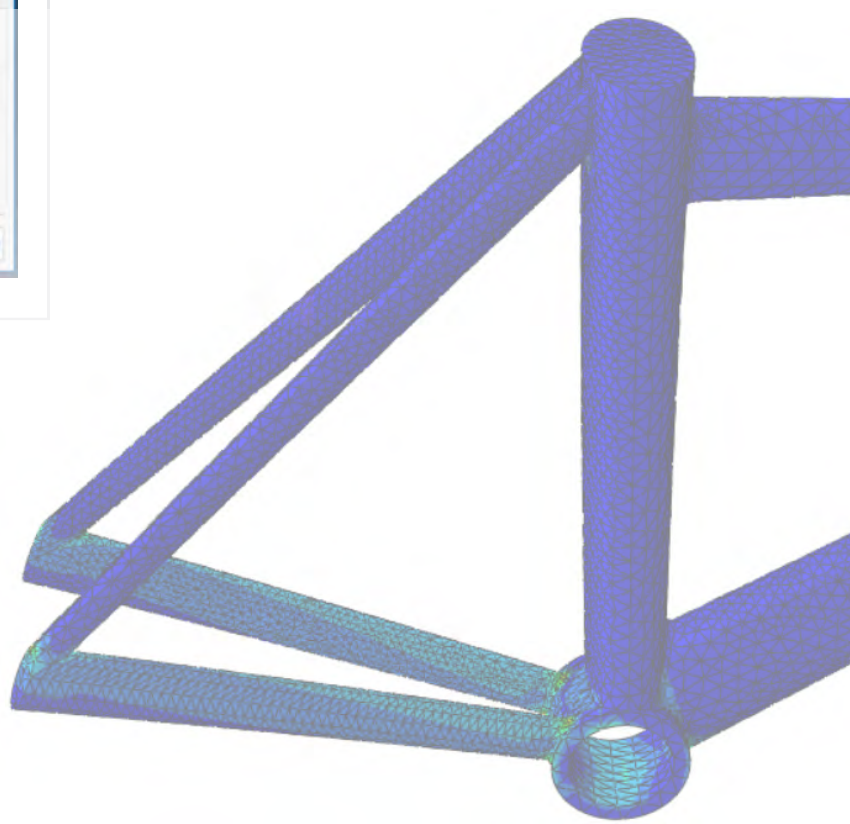
3

Once the user is satisfied with a microstructure, selecting “Create Abaqus Material” creates a user material in Abaqus which uses MultiMech to calculate the material’s response. This material can then be assigned to sections and composite layups like any other material inside Abaqus/CAE.

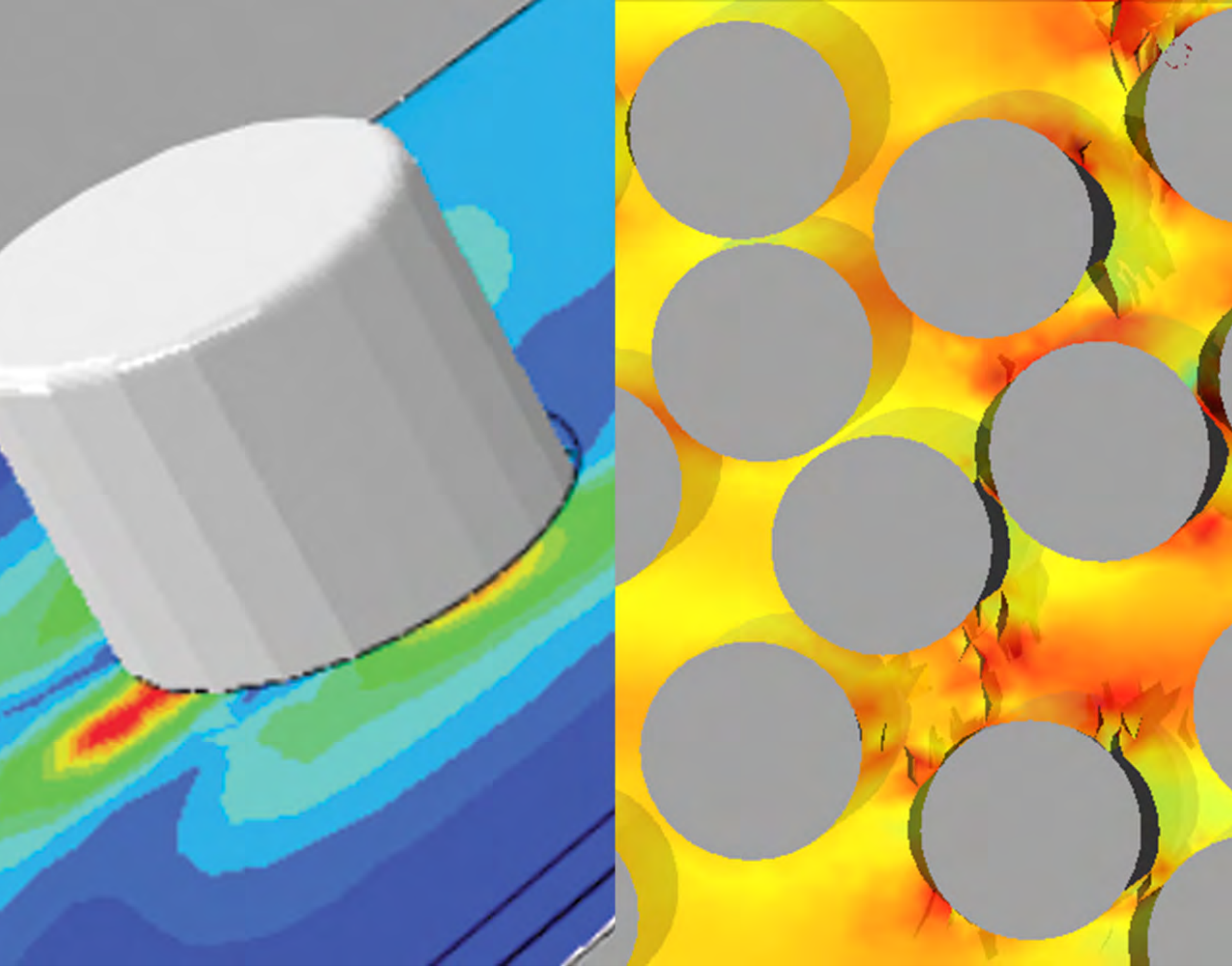


4

In the “Job” module the user can then create a MultiMech Job to define the main settings and activate the multiscale solution. Solving this Job will 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.



MultiMechanics



The MultiMech for Abaqus tool utilizes the extremely accurate and detailed FE^2 theory.

Using this theory, 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 every 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 due to the true damage criteria being represented. This method is truly representative due to the fact that microscale damage will lead to stiffness reduction in global scale elements, distributing stress concentration to different elements.

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

DE Technology for Optimal Design Engineering Editor's Pick of the Week

INNOVATOR OF THE YEAR FINALIST

TechConnect INNOVATION AWARD 2018

MultiMechanics

Virtual Testing Software For
The World's Most Advanced Materials

MultiMechanics.com

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