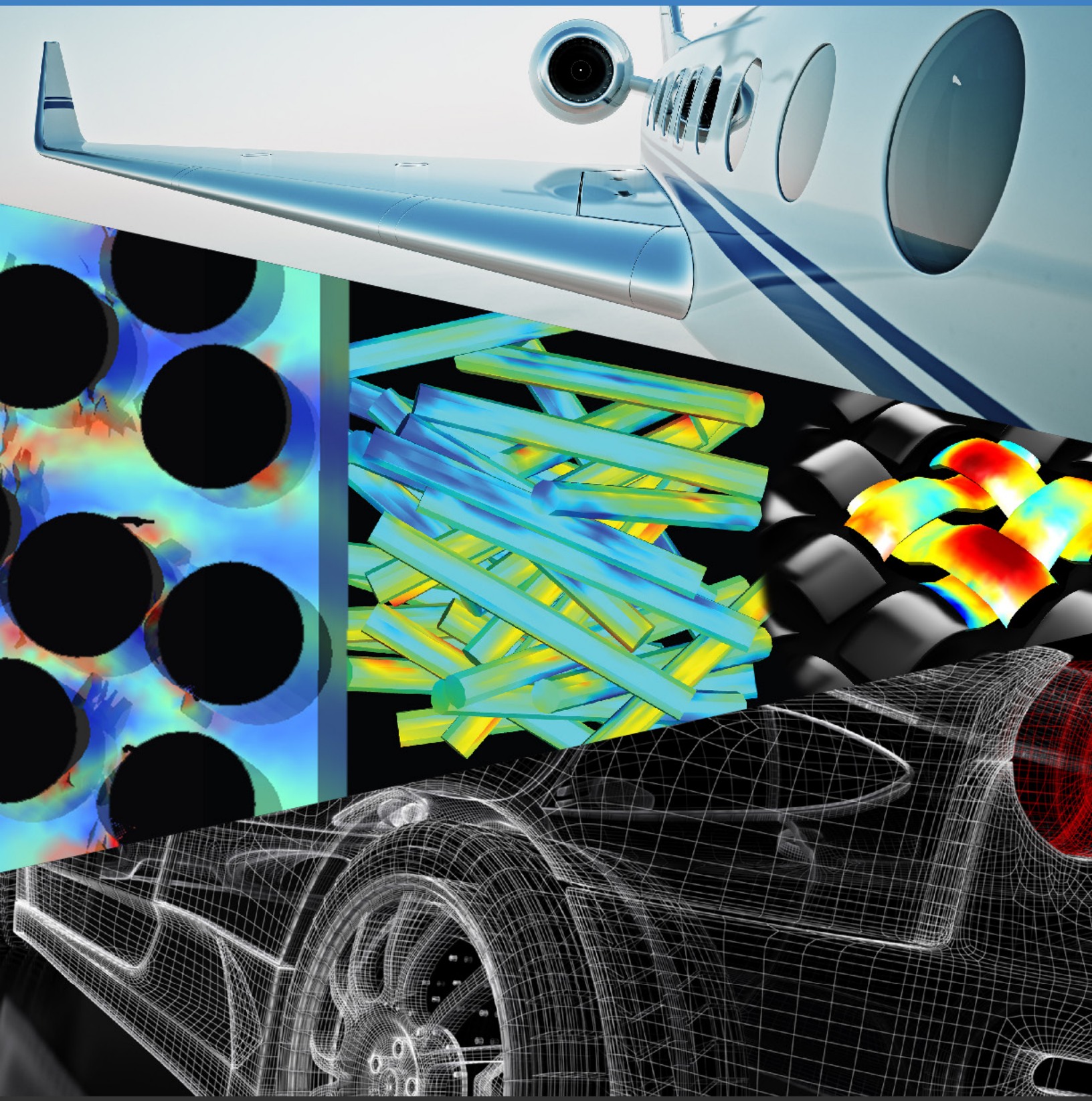
MultiMechanics

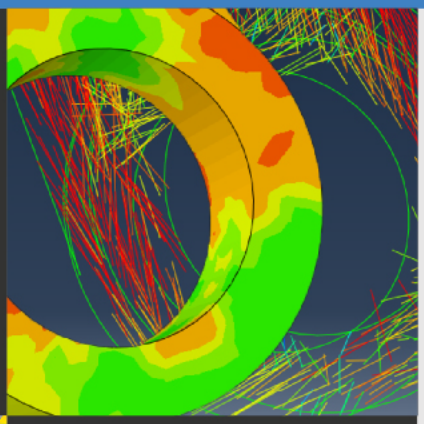
Virtual Testing Software for the World's Most Advanced Materials

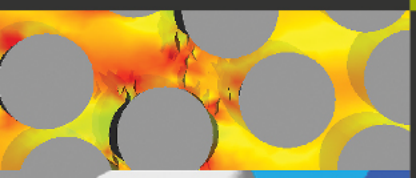


HOW WE HELP

Predict

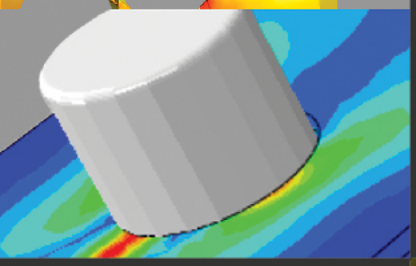
Account for manufacturing variability and imperfections to maximize product reliability





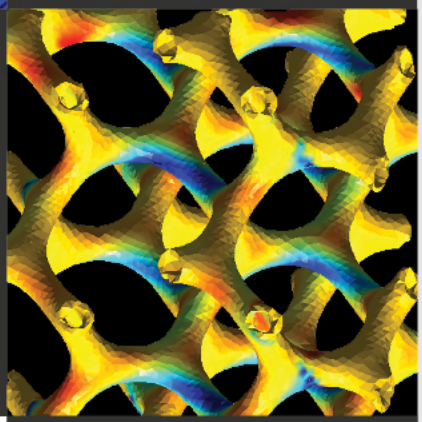
Zoom

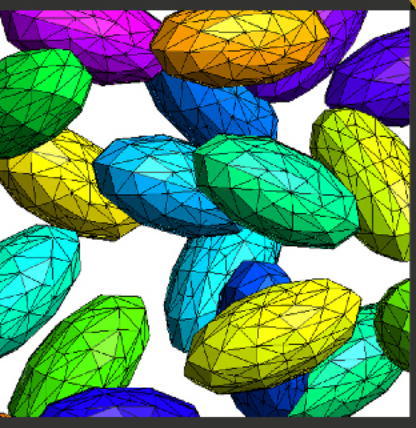
Zoom into the material microstructure to identify the root cause of failure and see how damage mechanisms affect structural performance



Optimize

Optimize the material microstructure for the most cost-efficient performance





Innovate

Create and test new and existing composites, including (but not limited to) woven, braided, particulate, and continuous fibers

Our flagship product **MultiMech** helps companies accelerate the product development lifecycle by virtually predicting failure in advanced materials at an unprecedented level of speed and accuracy.

Gain Faster Analysis Results

Our proprietary TRUE™ Multiscale technology, embedded within MultiMech, enables 1000x gains in efficiency. Computations that used to take days in HPC systems can now be solved in minutes, on a laptop.

Achieve Higher Accuracy

MultiMech produces virtual results that provide up to 99% accuracy, driving critical performance for engineers and helping to increase confidence in designs.

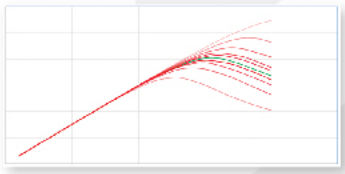
Reduce Time and Cost to Market

By reducing the amount of physical testing required to develop and certify new materials, MultiMechanics helps companies achieve 80%+ savings in time and cost.

TEST NEW MATERIALS IN MINUTES, ON A LAPTOP

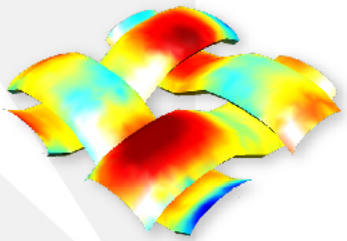
Automatic Microstructure Generation

MultiMech automatically generates the geometry and meshing of microstructural models. The only inputs required from the engineer are basic design variables.



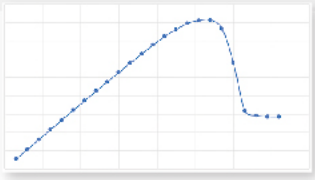
Optimization

Once the engineer provides the desired mechanical properties, MultiMech automatically changes the design variables until the right combination is achieved in order to meet the desired mechanical performance.



Virtual Testing

MultiMech quickly and easily applies various loading scenarios to the generated microstructure to give engineers insight into how the material will perform in different conditions.



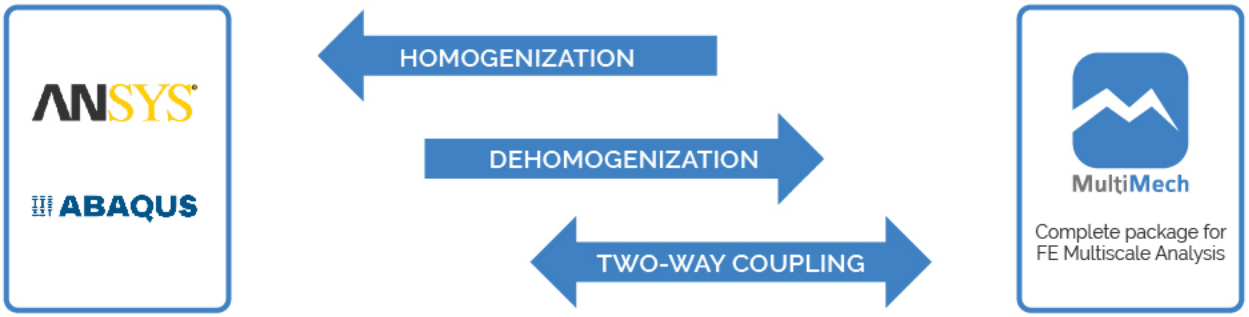
Composite Material Properties

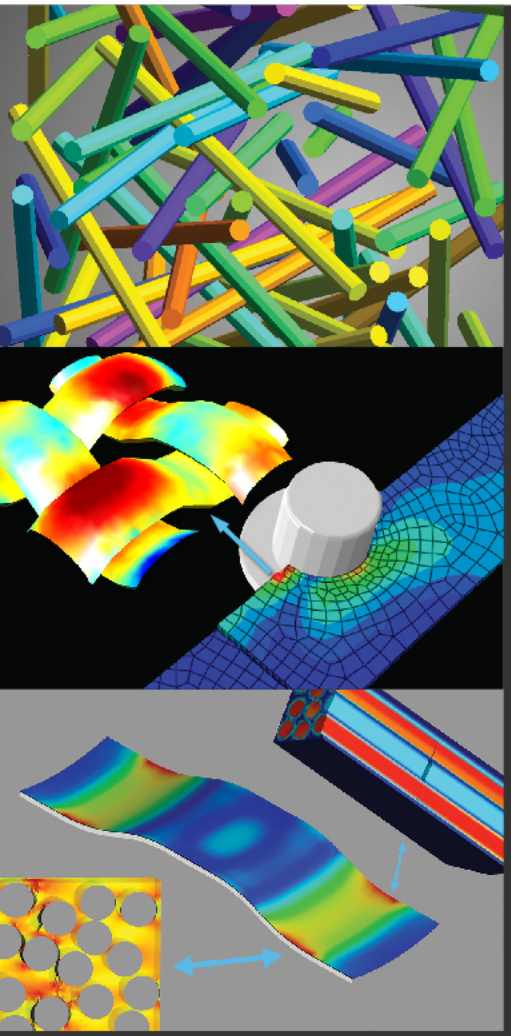
Engineers can quickly perform Design of Experiments to learn which design variables matter most for the design.

INTEGRATE PART, MATERIAL, AND PROCESS

Enhance Your Structural Analysis

Connect all elements of your analysis in order to quickly eliminate poor designs.





1. Homogenization

Use the test results obtained from MultiMech to run a quick, accurate analysis in your finite element solver. Gain insight into how the part will behave with the material you created in order to filter out inadequate designs. Our virtual testing capability allows engineers to design complex materials to efficiently represent linear and nonlinear behavior of materials in part simulation.

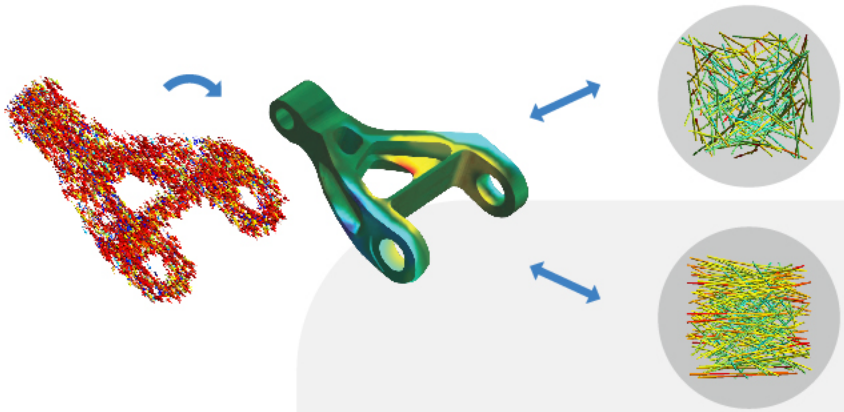
2. Dehomogenization

If the part being tested using homogenization undergoes significant deformation, MultiMech makes dehomogenization simple and intuitive. Simply select any element in your results and the software will automatically extract the strain in a given element. It will then apply that strain directly to the microstructural model, giving insight into whether a given level of deformation will result in microstructural damage.

3. Two-Way Coupling

For the most promising designs, MultiMech runs a full simulation both at the part and microstructural levels simultaneously. At every step, the updated properties of every RVE are sent back to their respective integration points. This feedback loop is critical in order to have a more accurate analysis. The load is transferred down from the part to each respective RVE.

MultiMech interacts with different data sources to import manufacturing/process variation data. It then converts the data into the appropriate microstructure models.



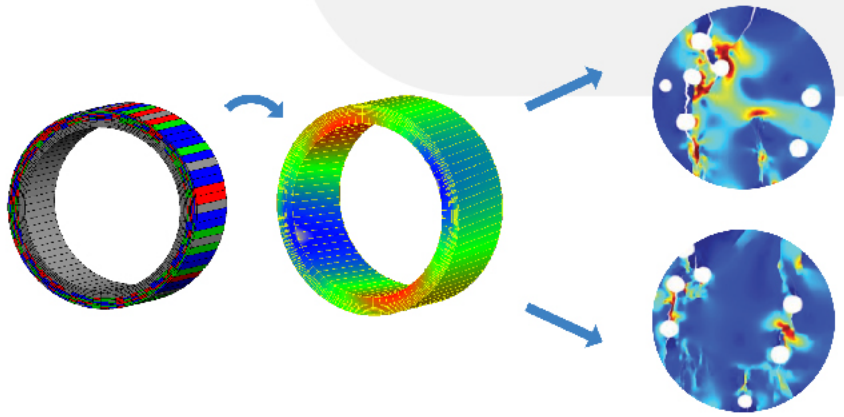
For injection-molded parts, MultiMech can import fiber orientation data from industry-standard injection molding simulation tools, such as Moldflow and Moldex3D. The software can then automatically map the fiber orientation data from the flow mesh to the structural mesh. Using this information, MultiMech generates multiple finite element microstructural models to their respective integration points of the structural mesh. This allows engineers to simulate "as manufactured" parts with a much greater level of accuracy, as they are able to take into account manufacturing-induced microstructural variations.

moldflow

Moldex3D

PROCESS VARIATION

MultiMech



In addition to fiber orientation, MultiMech also takes into account information about manufacturing-induced defects, such as voids, variation in volume fraction, resin pockets, and fiber misalignment. This data can come from multiple sources, including CT scans, can be mapped to a structural finite element model of the part, and can be used to generate microstructural models that can reflect the defects at a specific location. This enables engineers to simulate much more realistic parts, as they have accounted for the manufacturing-induced microstructural changes.

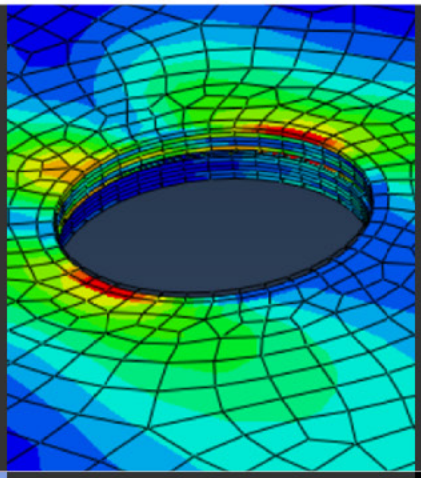
HOW IT WORKS

Packages

Use with your workflow or as a standalone tool:

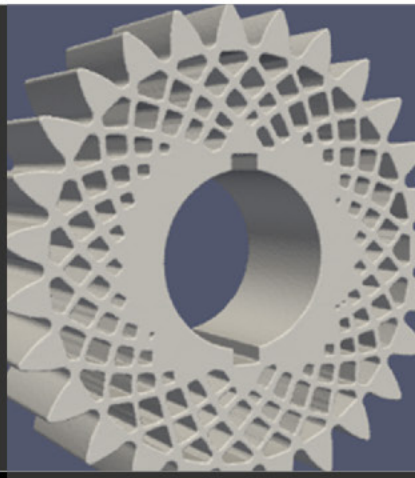
MultiMech for Abaqus

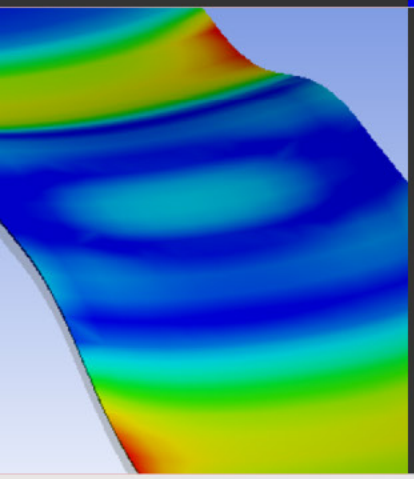
Intuitive TRUE™ Multiscale FE analyses within Abaqus



MultiMech API

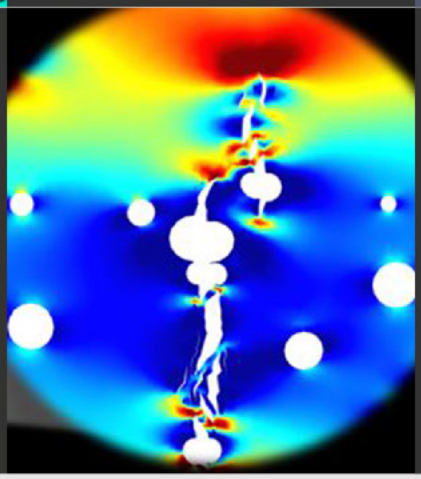
Extensible finite element API for advanced workflows and integrations with any CAE software, even if it is not FE-based





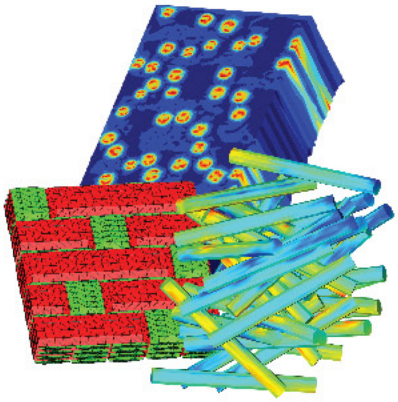
MultiMech for ANSYS

Intuitive TRUE™ Multiscale FE analyses within ANSYS/ Workbench



MultiMech Standalone

Full TRUE™ Multi-scale FE package integrated with open source pre/ post processors



TRUE
Multiscale Technology

MultiMech is powered by our proprietary TRUE™ Multiscale Technology, a data compression algorithm that reduces the time and cost of running computational analyses by up to 80%.

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.

Awards & Recognition for MultiMechanics:

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

SOLVAY

Nicolas Cudré Mauroux
Chief Technology Officer

"Our Composite Materials Global Business Unit carefully reviewed all modeling solutions and, by far, MultiMechanics provided the best results.

We are confident this software can accelerate innovation in complex materials and the penetration of composites in the automotive and aerospace industries.

The accuracy and speed afforded by MultiMechanics, and its efficient integration with commonly used commercial finite element software packages, is changing the way we develop new materials and interact with our customers."

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