

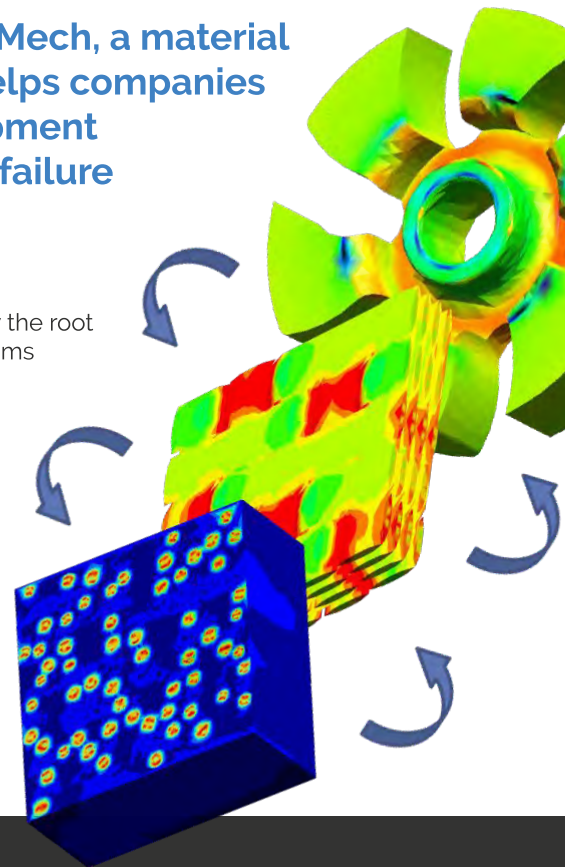
# MultiMech Product Overview

## GENERAL FEATURES

- **TRUE Multiscale™ Technology**
- **Automatic generation of microstructures**
- **Finite Element solver**
  - Linear and Nonlinear
  - Explicit and implicit
- **Progressive damage evolution**
- **Automatic crack initiation and growth**
- **Structural and material optimization**
- **Tube and pressure vessel optimization**
- **Reverse engineering via optimization**
- **Accurately model any multiphase material**
- **Full control over accuracy level and 1,000x faster than traditional FE2**
- **Integrated with Abaqus CAE, ANSYS/Workbench, Altair Hyperworks**

MultiMechanics develops MultiMech, a material simulation software tool that helps companies accelerate the product development lifecycle by virtually predicting failure in advanced materials.

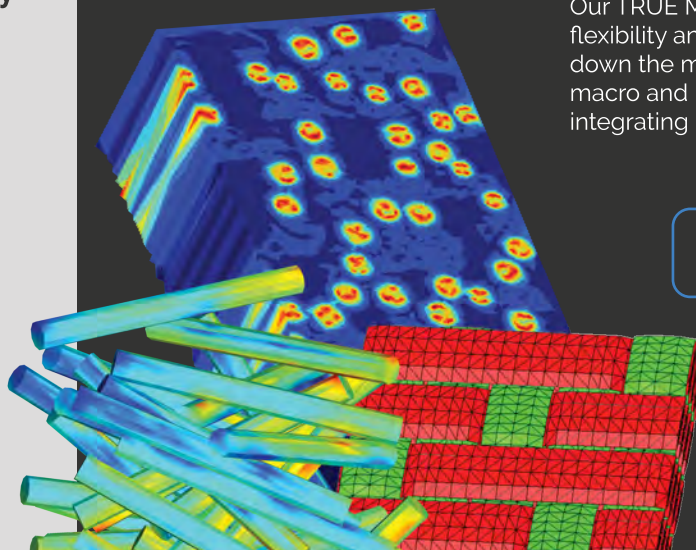
- Zoom into the material microstructure to identify the root cause of failure and see what damage mechanisms affect structural performance
- Account for manufacturing variability and imperfections to maximize product reliability
- Optimize the material microstructure for the most cost-efficient performance
- Create and test new and existing composites

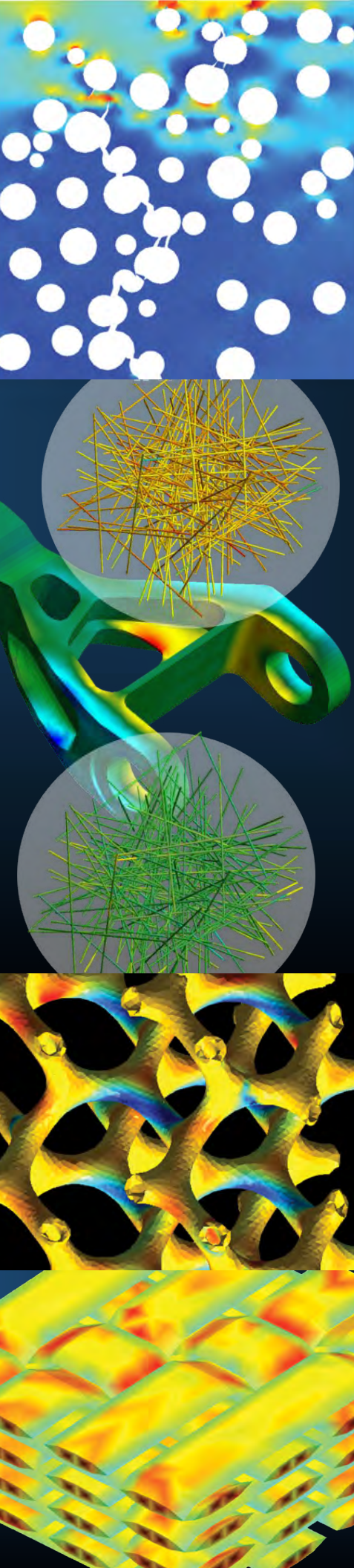


**MultiMech is an all-inclusive, Multiscale Finite Element software package for analysis of advanced materials.**

Our TRUE Multiscale™ technology extends the flexibility and robustness of Finite Element modeling down the microstructural level, strongly coupling the macro and micro mechanical response and integrating materials engineering to part design.

[Learn more inside ►](#)





“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.”

Nicolas Cudré-Mauroux, CTO, Solvay

## Key Features

- Automatic generation of complex, 3D Finite Element microstructures
- Import process-induced variation data and automatically convert it to microscale, including volume fraction and fiber orientation tensors to simulate injection-molded parts
- Perform implicit and explicit TRUE Multiscale™ finite element analysis
- Multiscale nesting scheme allows modeling of a virtually unlimited number of scales
- Advanced algorithm allows for accurate damage modeling, including multiscale transition of micro-cracks into macro-cracks
- Automatic insertion of cracks or cohesive zones
- Accurately model fiber-resin debonding and fiber rupture
- Model the effect of individual microscale constituents, including their interactions, volume fractions, spatial distribution and orientation, and fracture toughness, among other factors
- Fully parallelized and scalable on both standard multi-core desktops/laptops and HPC infrastructures
- Available as standalone software or as a plugin for most major CAE platforms
- Fast results due to our proprietary mathematical formulation and data compression algorithm

### Import:

- Abaqus (.inp) ASCII format
- Gmsh (.msh) ASCII format
- Injection molding data
- Moldex3D
- Moldflow (.XML)

### Supported platforms

- Windows (x86\_64)
- Linux (x86\_64)



**“When it comes to multiscale analysis and bottom-up prediction of complex materials, MultiMechanics has no peers.”**

**Matt Jackson, Senior Research Scientist, Solvay**

## Material Models

### ELASTICITY

- Isotropic thermo-elastic
- Orthotropic thermo-elastic
- Isotropic continuum - damage thermo-elastic

### VISCOELASTICITY

- Isotropic thermo-viscoelastic with aging
- Isotropic continuum damage thermo-viscoelastic with aging

### ELASTO-PLASTICITY

- Isotropic Von Mises thermo-plastic

### COHESIVE ZONE MODELS

- Linear decay
- Bi-linear
- Bi-linear rate-dependent
- Tvergaard
- Allen nonlinear viscoelastic
- Material interface (automatic crack/CZ insertion)

### CONTACT MODELS

- Elastic frictionless

### MULTISCALE MATERIAL MODELS

- Microscale RVE
- Microscale cohesive zone RVE

### DIFFUSION MODELS

- Isotropic Fourier
- Isotropic Fourier (thermoactive)

## Simulation Details

### ANALYSIS

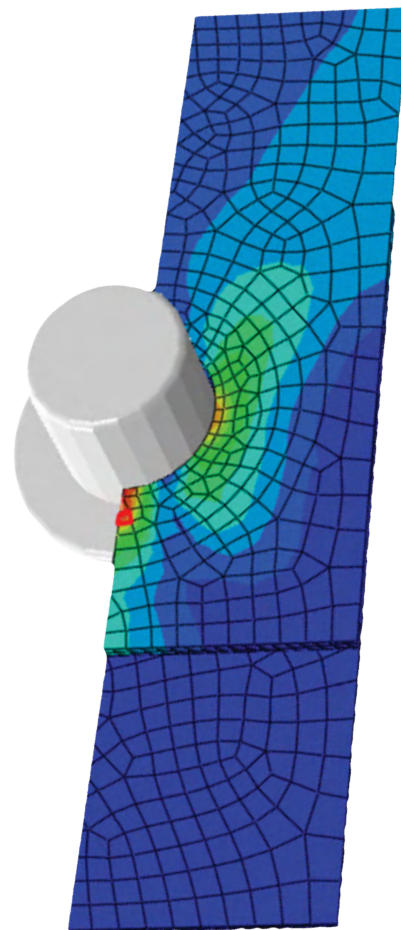
- Two-way coupled FE multiscale solver
- Quasi-static (implicit)
- Dynamic (explicit)
- Thermal diffusion
- Thermo-Chemo - Mechanical coupling

### AUTOMATIC GENERATION OF MICROSTRUCTURES

- Woven composites
- Continuous fibers
- Chopped fiber composites
- Particulates
- Voids

### PROGRESSIVE DAMAGE

- Multiscale failure models
- Automatic insertion of cracks (or cohesive zones)
- Automatic correction of interpenetrating interface elements
- Continuum damage models
- Element deletion
- Stiffness reduction
- Variety of failure envelopes



### SERVICES AVAILABLE:

- Consulting
- Training
- Custom software development

## 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](mailto:info@lanikasolutions.com) [www.lanikasolutions.com](http://www.lanikasolutions.com)