



SIEMENS

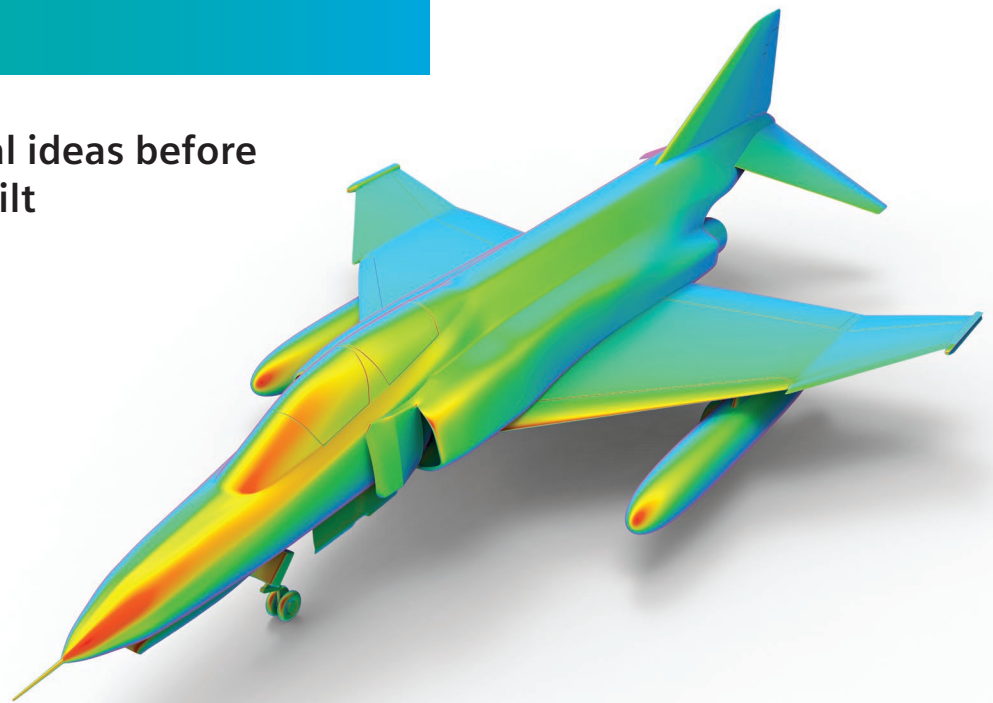
Ingenuity for life

Multimech for aerospace

Virtually test new material ideas before a physical prototype is built

Benefits

- Virtually test new material ideas before a physical prototype is built
- Enhance confidence the design will pass the physical testing and certification process
- Efficiently integrate with commonly used finite element packages
- Improve the way companies develop new materials and interact with customers
- Virtually optimize advanced materials



Summary

Multimech is a multiscale material modeling and simulation platform designed to address the challenges faced by companies involved in material design and manufacturing. It enables engineers to virtually test their new material ideas before a physical prototype is built, giving them much more confidence the design will pass the physical testing and certification process.

Multimech provides accuracy and speed, and it can be efficiently

integrated with commonly used finite element (FE) software packages, which changes the way companies develop new materials and interact with their customers.

Multimech enables users to model microstructural cracks and determine how they would affect the overall composite part. Using Multimech, manufacturers can see where a crack is developing, how the crack will change the material and how it will affect the final microstructure of the material.

Multimech for aerospace

How does Multimech fit into my workflow?

When simulating part performance in Abaqus, ANSYS or Simcenter™ Nastran® software, engineers can use the Multimech plugin to replace traditional, simplistic material models.

Multimech enables you to automatically create finite element microstructural models for a wide range of composite materials, including unidirectional and woven continuous fiber, chopped fiber, porous and particulate composites.

Progressive failure and microstructural damage can be evaluated to make important material and part design improvements.

Multimech allows companies to develop models and understand material behavior at multiple scales, including:

Constituent level

- Virtually certify materials, automatically generate microstructures and observe microstructural failure mechanisms such as fiber-resin debonding and cracking

Coupon level

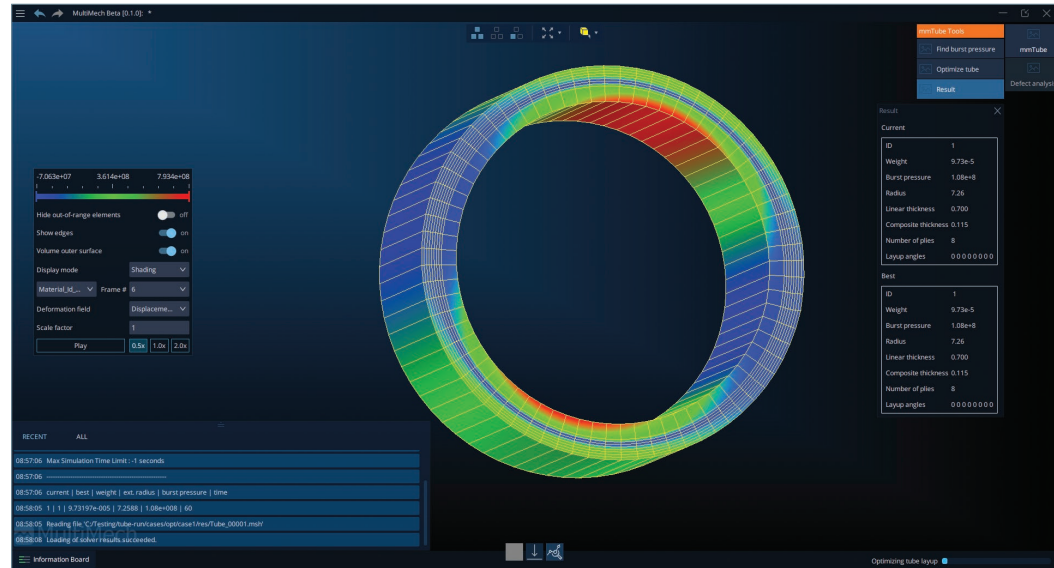
- Virtual material certification with accurate failure analysis of standardized test procedures

Component level

- Accurately predict failure and deformation in components
- Optimize geometry and material orientation

Full structure

- Use plugins to apply different loads to predict how the material will affect structural performance



How it works

Multimech is generic enough to handle custom models of materials, coupons and structures. It works as a standalone tool or can be integrated with standard FE solvers.

1. Automatically generate microstructures with complex material models

- a. Linear and nonlinear material response
 - i. Viscoelasticity
 - ii. Plasticity
 - iii. Automatic cracking
 - iv. Failure models

2. Import and export microstructures from database

3. Run simulations

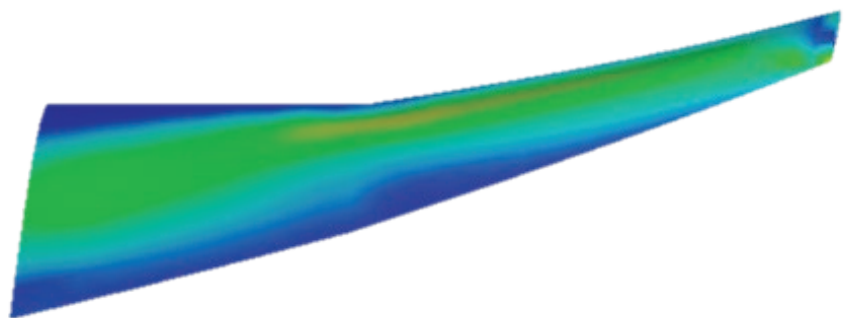
- a. Multiscale analysis
- b. Larger components
- c. Standardized coupon testing

4. Analyze results

- a. Microstructural damage
- b. Macrostructural failure

5. Optimize

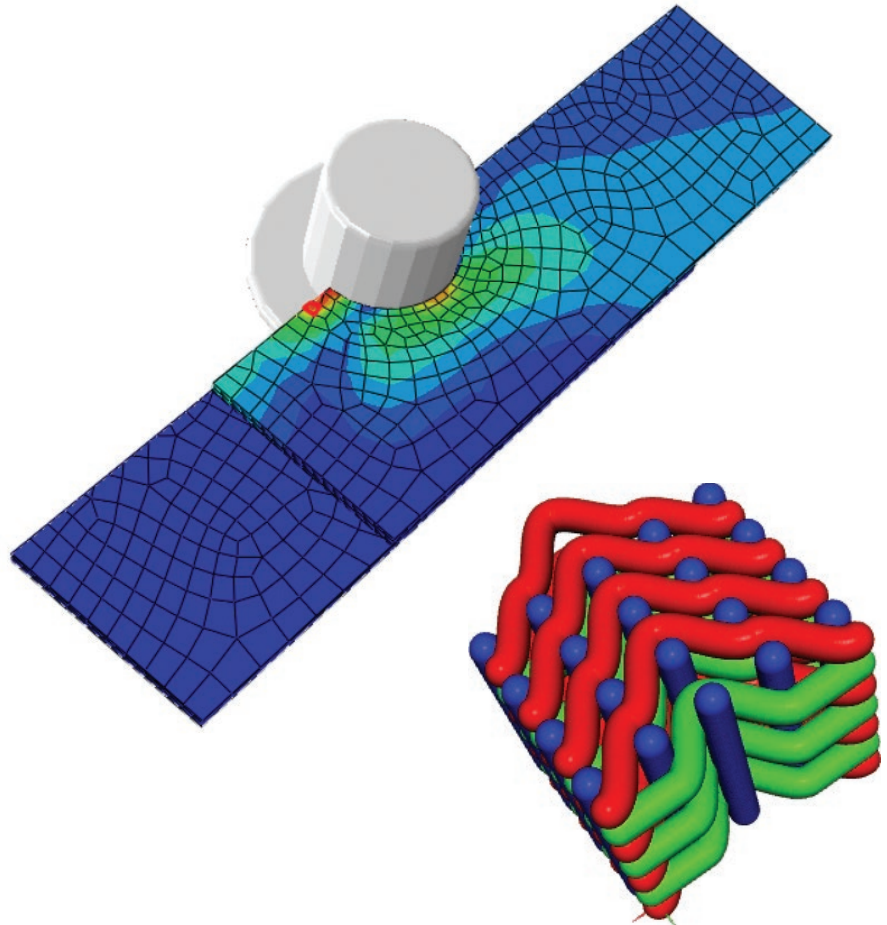
- a. Improve product design
- b. Meet desired target properties
- c. Minimize physical testing



Packages

Use Multimech with your current workflow or as a standalone tool:

- Multimech for Abaqus
 - Intuitive True Multiscale FE analyses within Abaqus
- Multimech for Ansys
 - Intuitive True Multiscale FE analyses within ANSYS Workbench
- Multimech for Simcenter 3D
 - Intuitive True Multiscale FE analyses within Simcenter Nastran
- Multimech API
 - Extensible finite element application programming interface (API) for advanced workflows and integrations with any computer-aided engineering (CAE) software, even if it is not FE-based
- Multimech standalone
 - Full True Multiscale FE package with open source pre/post processors



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