FREQUENTLY ASKED QUESTIONS

MULTIMECHANICS STANDALONE AND PLUGINS

1. WHAT ARE THE AVAILABLE APPLICATIONS FROM MULTIMECHANICS?

Application
MicroMech Microstructure Generator
MultiMech Virtual Testing Tool
MultiMech GUI
MultiMech for Abaqus / MultiMech for Ansys

2. WHAT IS THE FUNCTIONALITY PROVIDED BY THE ABOVE VERSIONS

MICROSTRUCTURE GENERATOR

- Automatic generation of microstructures
- Continuous and chopped fiber reinforced composites
- Woven composites
- Particulate composites
- Voids

GRAPHICAL USER INTERFACE

- Intuitive graphical user interface for single and multiscale problems
- Import mesh from various formats, including commercial and open source software

SOLVER FEATURES – MULTISCALE AND VIRTUAL TESTING

- Single scale solver
- Fully parallelized multiscale solver
- Tools for optimizing imported meshes
- Automatic insertion of cracks or cohesive zones
- Automatic correction of interpenetrating interface elements
- Quasi-static and dynamic/impact problems
- All material models

- Contact models
- Local and remote job execution
- Export results into various formats, including commercial and open source software

PLUGIN FEATURES

- Tight UMAT integration with Abaqus and Ansys where MultiMech RVE acts as a native material model
- Integration with Fiber Orientation output, including mapping between flow and structural mesh
- Support for Continuum and Shell elements
- Post-Processing entirely in CAE tool
- Fully Parallelized

5. WHAT ARE THE AVAILABLE HARDWARE PLATFORMS FOR MULTIMECH?

MultiMech is available for Windows and Linux as summarized below:

Application	Platform Support
MultiMech	Windows (64 bit) on X86-64 Linux (RHEL 6, 64 bit) on X86-64

6. WHO DO I CONTACT FOR PRE-SALES AND POST-SALES TECHNICAL SUPPORT?

Technical support contacts are summarized below:

ISV support contacts

Support Territory	Support contact	Email address	Phone number
Indian subcontinent	Venkatesulu	support@lanikasolutions.com	+91-80-2548 4844

WHAT IS MULTIMECH?

MultiMech is a software suite designed for structural analysis of advanced materials. It's true strength lies in its ability to perform fast and accurate multi-scale simulations on complex heterogeneous composites. The software itself is standalone and consists of a geometry and

meshing software (pre-processor), material definition software, multi-scale implicit/explicit solver, and post-processor viewing software.

WHAT IS MULTIMECH FOR ABAQUS / MULTIMECH FOR ANSYS

These are plugins for Abaqus/Standard and Ansys Mechanical, respectively. They let engineers use MultiMech's breakthrough microstructure modeling and multiscale analysis techniques completely within these standard CAE platforms.

This means that engineers no longer need to generate material cards to capture the unique behavior of their composite microstructure. Instead, they can use the raw-material properties of the constituents to derive the composite properties.

WHAT VERSIONS OF ABAQUS ARE SUPPORTED?

ABQ6.13 and ABQ6.14 are supported by the plugin. Note that depending on which version of Abaqus you are using, you will need a separate version of the plugin.

HOW DOES MULTIMECH MODEL DEFECTS WITHIN A COMPOSITES?

This is absolutely no problem for MultiMech. We have the most advanced crack algorithm in market, as well as simpler element deletion and continuum damage models. Besides, our mathematical approach allows us to calculate homogenized properties with exact expressions even in case of explicit crack growth - the only approximation is the FE discretization itself - which is commonly accepted in industry and easy to control by refining the mesh.

WHAT IS THE PHYSICS BEHIND MODELING DAMAGE?

Our solver handles multiple ways of modeling damage: automatic insertion of cohesive zones, element deletion, continuum damage models, etc. We are also open implement new options based on clients needs. In order to learn more about the physics behind them, please check the following attachments: Material Modeling - MultiMechanics.pdf Failure Modeling - MultiMechanics.pdf

CAN YOUR SOLVER PREDICT MATERIAL FAILURE?

Yes, MultiMech is able to accurately predict both local and global scale failure, with good physical correlation. You can also replay your simulation, seeing exactly which element failed and why.

WHAT TYPE OF ANALYSIS DOES MULTIMECH SUPPORT?

MultiMech has two different solvers, one for explicit and one for implicit analysis. You simply need to specify the type of simulation and MultiMech does the rest

WHAT ARE SOME EXAMPLES OF SOPHISTICATED COMPOSITES THAT MULTIMECH IS ABLE TO MODEL.

MultiMech's automatic microstructure generator can easily, and quickly generate a wide range of material microstructures, including but not limited to: 1. continuous fiber reinforced polymer matrix composites and ceramic matrix composites 2. unidirectional prepregs, 2D/3D woven fabrics with 3 to 4 distinct constituents in the phase microstructure.

DO YOU OFFER A MICROMECHANICS TOOLS FOR CONSTITUENT LEVEL STRESSES/STRAINS AND ELASTIC PROPERTIES

MultiMech has an easy-to-use Material Virtual Testing feature that automatically builds and tests microstructural models, resulting in, but not limited to, "constituent level stresses/strains and elastic properties"

DO YOU OFFER MICROSTRUCTURE LEVEL DAMAGE MODELING FOR MATERIAL RESPONSE AND FAILURE PREDICTION?

The same Material Virtual Testing feature be used to predict failure, and can model multiple damage mechanisms simultaneously, such as fiber breaks, matrix cracking, fiber-matrix debonding; The constituents can also have nonlinear behavior such as plasticity, and viscoelasticity.

WHAT KIND OF FAILURE MODEL, WHICH ARE AVAILABLE IN MULTIMECH, IS SUITABLE FOR BRITTLE, DUCTILE, AND COMPOSITE FRACTURE?

We support several different composite failure mechanisms (Tsai-Wu, Hashin), but also support constituent level failure material models like continuum damage, visco-elasticity, plasticity, etc. and failure models like MPS and stiffness reduction. Our Crack modeling algorithm, which we would recommend for modeling damage in brittle materials has a number of sophisticated methods for modeling the cracking characteristics of a material, such as the visco-elastic cohesive behavior and Weibull distribution of material strengths.

DO YOU OFFER MULTI-SCALE INTEGRATION CAPABILITY AND GLOBAL RESPONSE?

MultiMech has a unique, proprietary, TRUE Multiscale approach that allows efficient, two-way coupled link between global and local scale structures. Examples of this feature can be found in: https://www.youtube.com/watch?v=_oSq4OvlZoE
https://www.youtube.com/watch?v=h_XBgr2pZsc

DO YOU HAVE BOTH 2D AND 3D ELEMENT MODELING CAPABILITIES

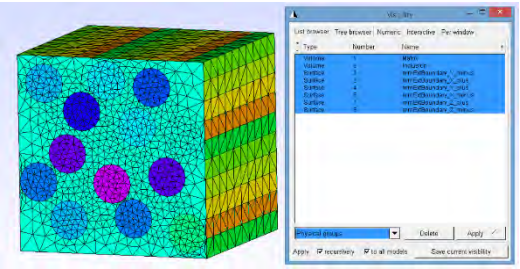
Yes - our tool is equally adept at modeling both 2d and 3d elements.

DOES THIS ONLY GENERATE TET MESH, OR CAN IT GENERATE HEX MESH AS WELL?

Currently, with our mesh generator it's only tet mesh. Hex mesh capabilities will be added later.

CAN YOU PROVIDE MORE DETAIL ABOUT THE MICROSCALE RVE?

We generate non-idealized, 2d/3d non-periodic meshes automatically for UD, Chopped, Woven, and Voided composite microstructures. The Meshes are in the GMSH .msh ascii format and are easily modified. Typically they are tet elements, but we can mesh with hex elements as well. The geometric inputs for these composites are all easily modified (diameters, volume fractions, distributions, etc.)



Each mesh includes 1. Physical groups for all constituents, 2. Boundary surfaces for all 6 faces. 3. Surfaces elements between all elements (for cohesive zone modeling)

IS THERE AN OBVIOUS LIMITATION TO THE VOLUME FRACTION YOU CAN SUPPORT FOR CHOPPED FIBER?

Limitation is close to what you see in practice For example on chopped fiber, you can see volume fraction around 30% If you want higher volume fraction, the intensity goes up Chopped fiber, collision detection is very complex Woven is quicker

HOW LONG DOES IT TAKE TO SIMULATE A COMPOSITE MATERIAL, USING 1 OR 2 DIFFERENT LENGTH SCALES?

As a rule of thumb, we say that it takes 1 sec per step per 5k unique local elements (per core). (That's a mouthful, I know).

So for instance, if you have a part with 100 global elements, and each has a uniquely evolving RVE, where each RVE is 5,000 elements, and you are running on a single core, then you are adding 100 seconds of overhead computation to each simulation step. If you have 100 cores, then you are adding 1 second of computational overhead per step.

So there's some engineering required to reduce the number of "uniquely evolving RVES" and to reduce the number of degrees of freedom within those RVEs.

HOW CAN WE OBTAIN THE SIMULATION PARAMETER, COHESIVE LAW, EXPERIMENTALLY?

Typically we start with rough data from the literature and work backwards. As for the bottom-up determination, let me confer with some colleagues and get you with a thorough answer.

Can the simulations show the change in fracture toughness with the change in composition?

Initially, you may need to hand calculate this using basic experimental outputs at the element level like Stress, Strain, C-matrix, Material Modulus, etc. Down the road we could automate the calculation of this.

CAN THE SIMULATIONS SHOW THE STOCHASTIC BEHAVIOR OF FRACTURE STRENGTH?

You can model variations in the fiber diameter, fiber location, fiber orientation (if chopped fiber), coating thickness, fiber/matrix/interface strengths, and more. For instance, we can simulate the Weibull distribution of the neat material's failure criteria, and/or the stochastic variation in pore size / location, which should result in stochastic fracture strengths.

WHAT KIND OF COMPUTER POWER WOULD BE OK TO RUN COMPLEX SIMULATIONS? DOES PARALLELIZATION WORK? IF WORKS, DOES IT SCALE FOR MORE THAN 64 CORES?

Most results can be obtained in a single workstation. Of course, if you have very refined macro and microscale meshes, you'll obviously need more memory. The good news is that MultiMech is fully parallelized so you can run simulations in HPC clusters if needed.

- 1scale virtual testing: - 16GB 4Core laptop
- 3d multiscale cases with isolated damage - 16 core 32GB workstation
- Complex 3d part, scattered damage - 64+GB Memory 32+cores recommended.

DO I NEED TO REVERSE ENGINEER FIBER/MATRIX MATERIAL PROPERTIES?

MultiMech is a predictive physics-based tool. We've done many studies for our clients where we enter data from manufacturer's data sheet. In our opinion, reverse engineering microscale properties is only valid if the in-situ and lab curing conditions are very different, otherwise it sounds like you're just curve fitting your model, and the model won't give good results if things like loading rate changes.

WHAT PROPERTIES DO YOU REQUIRE TO MODEL CRACKS EXPLICITLY?

MultiMech offers a variety of ways to model damage at both part and microstructural levels - for example: continuum damage models, element deletion, cohesive zones (including viscoelastic CZ), cracks, and so forth. Each require different parameters, but to model cracks in MultiMech you only need to enter the tension and shear strengths of the matrix, fiber and fiber-matrix interface.

WHY IS MULTIMECH'S MULTISCALE TECHNOLOGY DIFFERENT?

We have the most accurate coupling of all available technologies in market. We make no simplifications on deformation modes. The only assumption is on the boundary condition applied at local scale, which are either linear displacement or uniform traction. Further we do not lose accuracy, because our algorithm is sufficiently fast. We can solve true multiscale analysis with automatic insertion of cracks in few minutes and virtual tests in less than a minute.

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