Digimat 2023.1

MF User's Guide

HEXAGON

Americas
5161 California Ave. Suite 200
University Research Park
Irvine, CA 92617
Telephone: (714) 540-8900
Email: americas.contact@hexagon.com

Europe, Middle East, Africa
Am Moosfeld 13
81829 Munich, Germany
Telephone: (49) 89 431 98 70
Email: info.europe@hexagon.com

Japan
KANDA SQUARE 16F
2-2-1 Kanda Nishikicho, Chiyoda-ku
1-Chome, Shinjuku-Ku
Tokyo 101-0054, Japan
Telephone: (81)(3) 6275 0870
Email: MSCJ.Market@hexagon.com

Asia-Pacific
100 Beach Road
#16-05 Shaw Tower
Singapore 189702
Telephone: 65-6272-0082
Email: APAC.Contact@hexagon.com

Worldwide Web
www.mscsoftware.com, www.hexagon.com

Support
https://simcompanion.hexagon.com

Disclaimer

March 23, 2023

DIGI:V2023.1:Z:Z:Z:DC-UG-PDF

Documentation Feedback

At Hexagon, we strive to produce the highest quality documentation and welcome your feedback. If you have comments or suggestions about our documentation, write to us.

Please include the following information with your feedback:

- Document name
- Release/Version number
- Chapter/Section name
- Topic title (for Online Help)
- Brief description of the content (for example, incomplete/incorrect information, grammatical errors, information that requires clarification or more details and so on).
- Your suggestions for correcting/improving documentation

You may also provide your feedback about Hexagon documentation by taking a short 5-minute survey.

Note:	The above mentioned e-mail address is only for providing documentation specific feedback. If you have any technical problems, issues, or queries, please contact Technical Support.

Contents

Digimat MF User's Guide

8 RVE

9 Failure

14 Known Limitations

References

Preface

- About This Guide
- Purpose of This Guide
- Typographical Conventions
- Technical Support
- Accessing Digimat Documentation
- Internet Resources

About This Guide

This Guide is the *Digimat MF User Guide*. It contains information about the standard usage of Digimat-MF and it's interface with 3rd party software tools.

Purpose of This Guide

This guide explains the procedure for standard usage of Digimat-MF. The purpose of this guide is to provide user:

- An overview of Digimat-MF's capabilities around homogenization setup and microstructure generation
- Details of the theory behind Mean-Field homogenization in Digimat-MF
- Details of the GUI of Digimat-MF and explain standard usage workflows
- Details of the different material models and microstructures supported in the RVEs created in Digimat-MF
- Details of the various failure methods and equations underlying them
- Information about how to post-process homogenization results

The information in this manual is both theoretical and descriptive. You will find some techniques discussed in detail.

Typographical Conventions

This section provides a brief overview of the typographical conventions used in the document to help the user better follow the Digimat documentation.

This section describes some syntax that will help you in understanding text in the various chapters and thus in facilitating your learning process. It contains stylistic conventions to denote user action, to emphasize particular aspects of Digimat to signal other differences within the text.

LM Sans 10	Body and general text
Courier New	▪ Represents command-line options of Digimat. ▪ Directory names and paths ▪ File names and Paths ▪ Linux terminal script `Example: lmreread -c <parent>/msc/MSC.Licensing/licenses/license.dat`

Bold Text	▪ Highlights ▪ Dialog box names ▪ Buttons ▪ Menus ▪ User inputs ▪ The commands/user inputs for all descriptions related to terminal commands. ▪ Default values Example: `[root@vm-tmrhel73 MSC]#` **./msc_licensing_helium_linux64.bin**
HelveticaNeue LT Pro Cn 57	▪ Hyperlinks ▪ Weblinks Example: Chapter 10: Loadings
Italic Text	Represents references to books. Example: *Digimat AM User's Guide*

Technical Support

For technical support phone numbers and contact information, please visit:
https://simcompanion.hexagon.com/customers/s/article/support-contact-information-kb8019304

Support Center

https://simcompanion.hexagon.com

The Support Center provides technical articles, frequently asked questions, and documentation from a single location.

Accessing Digimat Documentation

This section describes how to access the Digimat documentation outside of Digimat. Digimat documentation is available through PDF files. The PDF files can be obtained from the following sources:

- Digimat documentation installer
- SimCompanion
- Combined documentation

The PDF documentation files are appropriate for viewing and printing with Adobe Acrobat Reader (version 10.1.4 or higher), which is available for most Windows and Linux systems. These files are identified by a .pdf suffix in their file names.

Downloading the PDF Documentation Files

You can download the PDF documentation from SimCompanion (https://simcompanion.hexagon.com).

Navigating the PDF Files

For the purpose of easier online document navigation, the PDF files contain hyperlinks in the table of contents and index. In addition, links to other guides, hyperlinks to all cross-references to chapters, sections, figures, tables, bibliography, and index entries have been applied.

To open the cross reference to other guides in a new window, you can make following changes to your Adobe Reader settings:

1. Click **Edit** -> **Preferences**.
2. Select **Documents.**
3. Uncheck **Open cross-document links in same window** option.
4. Click **OK**.

Printing the PDF Files

Adobe Acrobat PDF files are provided for printing all or part of the manuals. You can select the paper size to which you are printing in Adobe Acrobat Reader by doing the following:

1. Click **File**.
2. Select the **Print**.... option. The **Print** dialog box is displayed.
3. Select **Page Setup**....
4. Choose the required paper size in the **Page Setup** menu.

The PDF files are recommended when printing long sections since the printout will have a higher quality.

If the page is too large to fit on your paper size, you can reduce it by doing the following:

1. Select the **File** -> **Print**.
2. Under **Page Scaling,** choose the **Shrink to Printable Area** option.

Internet Resources

Hexagon (www.hexagonmi.com/mscsoftware)

Hexagon corporate site with information on the latest events, products, and services for the CAD/CAE/CAM marketplace.

Hexagon Download Center (https://mscsoftware.subscribenet.com/)

1 Overview

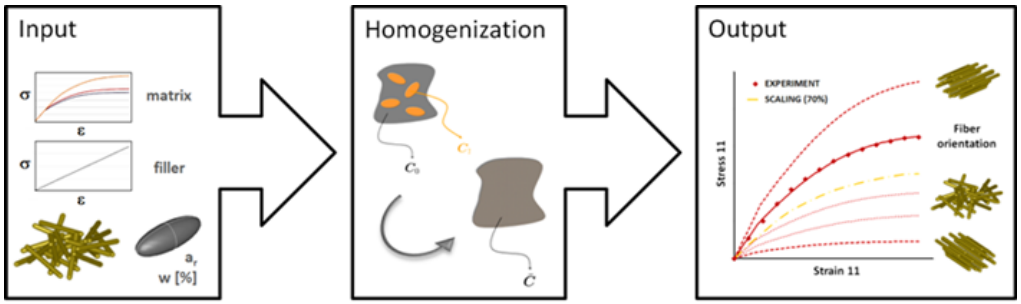
- General Description
- Homogenization Methods
- Micro (per-phase) Material Models
- Microstructure (Composite) Morphology
- Failure Indicators
- Loading

General Description

Digimat is the state-of-the-art linear and nonlinear multi-scale material modeling software suite from Hexagon.

Digimat software enables the prediction of the constitutive behavior of heterogeneous and/or anisotropic materials such as Polymer Matrix Composites (PMC), Rubber Matrix Composites (RMC) and Metal Matrix Composites (MMC).

Digimat-MF general workflow is as follows:



Per-phase material properties + microstructure information

State-of-the-art homogenization methods

DIGIMAT material model sensitive to composite microstructure

One typical and commonly used application of Digimat-MF is in the engineering plastics area where Digimat is used to model the linear and nonlinear material behavior of glass-reinforced thermo-plastic injection molded parts. In such a case Digimat will typically take into account the fiber orientation predicted by injection molding software and act as the micro-mechanical material model within the structural finite element analysis software.

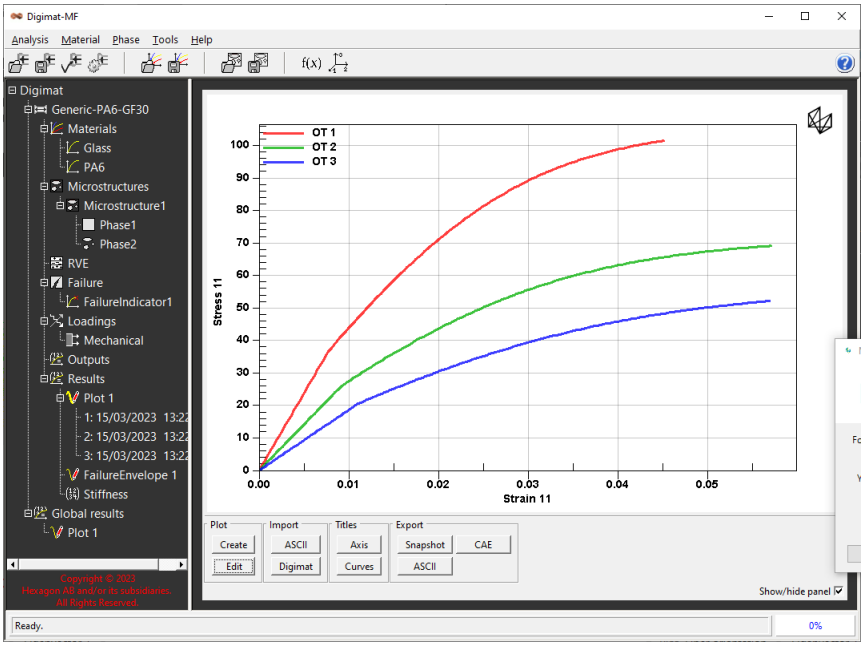


Figure 1-1 Digimat-MF graphical user interface (GUI).

For more information,

- On **known limitations** of Digimat-MF, click here.
- On **guidelines** for the usage of Digimat-MF, click here.
- For **examples & tutorials**: refer Examples and Tutorials guide.

The description of the software major capabilities are provided in the following sections. Some capabilities may be temporarily restricted to particular types of material combinations, microstructures or loading conditions. Please refer to the detailed sections in this user manual or contact digimat.support@hexagon.com for a demonstration or for more detailed information concerning the Digimat software suite.

Digimat-MF is the mean-field homogenization module of the Digimat software suite. It uses Eshelby-based semi-analytical mean-field homogenization approaches and an analytical description of the material in order to compute the thermo-mechanical, thermal or electrical properties of a composite as a function of its microstructure morphology, that is, inclusion shape, orientation, volume/mass fraction, and micro, that is, per-phase, material behavior.

The following capabilities are available in the current version of Digimat-MF.

Homogenization Methods

- Mori-Tanaka & Double inclusion homogenization schemes

- Multi-inclusion homogenization: multi-step and multi-level methods
- For nonlinear composite materials
  - First and second order homogenization schemes
  - Discrete Affine method
  - Interaction law scheme

Micro (per-phase) Material Models

- Linear thermo-elastic: large rotations (small strain)
  - Isotropic / transversely isotropic / orthotropic / anisotropic
  - Constant or temperature-dependent material properties
- Thermo-Elasto-plastic: J_2-plasticity
  - Isotropic hardening: power law / exponential law / exponential-linear law
  - Small deformation with small or large rotations
- Pressure-dependent Elasto-plastic: generalized Drucker-Prager
  - Isotropic hardening: power law / exponential law / exponential-linear law
  - Small deformation with small or large rotations
- Elasto-plastic with damage: Lemaitre-Chaboche damage model:
  - Plasticity model: J_2-plasticity
- Cyclic Elasto-Plastic: Chaboche-Marquis: Kinematic hardening: linear or linear with restoration
- Thermo-Linear viscoelastic
  - Isotropic
  - Constant or temperature-dependent material properties
  - Three shift functions: WLF / Arrhenius / Piecewise linear
- Thermo-Elasto-viscoplastic: J_2-plasticity
  - Isotropic hardening: power law / exponential law / exponential-linear law
  - Viscoplastic models: Initial yield Norton law / current yield Norton law / power law / Prandtl law (hyperbolic sinus) / time law
  - Constant or temperature-dependent material properties
  - Small deformation with small or large rotations
- Viscoelastic-Viscoplastic: J_2-plasticity
  - Isotropic hardening: power law / exponential law / exponential-linear law
  - Viscoplastic models: Initial yield Norton law / current yield Norton law / power law / Prandtl law (hyperbolic sinus) / time law

    - Small deformation with small or large rotations
- Strain rate elastoplastic: J_2 plasticity
    - Isotropic hardening: exponential-linear law
- Hyperelastic (finite strain)
    - Neo-Hookean: compressible or incompressible.
    - Mooney-Rivlin: compressible or incompressible
    - Ogden: compressible or incompressible
    - Swanson: compressible or incompressible
    - Störackers: highly compressible
- Crystal plasticity
    - Crystal Symmetry: FCC, BCC, HCP
    - Isotropic hardening model: Voce, Asaro-Needleman
- Leonov-EGP model (finite strain)
- Linear Fourier model (thermal analysis)
- Linear Ohm model (electrical analysis)
- Fourier Thermoelastic (Coupled thermal mechanical)
- Fourier Thermoviscoelastic (Coupled thermal mechanical)

Microstructure (Composite) Morphology

- RVE type: single microstructure or multilayer
- Microstructure type: Generic, Fabric, Lattice, Sheet Molding Compound or Metal
- Number of reinforcing phases: $\mathrm{N} \geq 1$
- Inclusion shape: Ellipsoids of revolution (platelets, spheres, fibers)
- Aspect ratio distribution
- Reinforcement orientation
    - Constant (fully oriented)
    - Random (2D or 3D)
    - Non-uniform (defined by an orientation tensor)
- Percolation model for electrical and thermal analyses

Failure Indicators

- Applied on micro (phase), pseudo-grain and/or macro (composite) level in global axes, local axes or tensor's principal axes

- Strain rate dependency in failure criteria can be defined
- Failure models
  - Maximum component (stress-based or strain-based)
  - Tsai-Hill 2D & 3D & 3D transversely isotropic (stress-based or strain-based)
  - Azzi-Tsai-Hill 2D
  - Tsai-Wu 2D (stress-based or strain-based) & 3D transversely isotropic (stress-based or strain-based) & 3D orthotropic (stress-based or strain-based) & 3D (stress-based only)
  - Multi-component 2D & 3D orthotropic
  - Hashin-Rotem 2D
  - Hashin 2D & 3D
  - SIFT
  - Christensen
  - Accumulated plastic strain
  - Camanho
  - User-defined
- Progressive failure models
  - Matzenmiller 2D
  - Matzenmiller 3D
- Failure models for high cycle fatigue analysis
  - pseudo grain model
    - Tsail-Hill 3D transversely isotropic
    - Modified Gerber
    - Energy based
  - matrix damage model

Loading

- Thermo-Mechanical
- Multi-axial stress, strain, harmonic stress or harmonic strain
- Monotonic, cyclic or user-defined time history
- Thermal loading: monotonic temperature gradient
- Electrical loading: monotonic voltage gradient

2 Mean-field Homogenization Theory

- Heterogeneous Composite Materials
- Micro-macro (two-scale) Approach
- General Averaging Results
- Homogenization and Scale-transition Methods
- Mean-field Homogenization (MFH)

Heterogeneous Composite Materials

We consider heterogeneous materials whose microstructure consists of a matrix material and multiple phases of so-called *inclusions*, which can be short fibers, platelets, micro-cavities or micro-cracks (see Figure 2-1).

There are several examples of such composite materials, such as:

- thermoplastic polymers reinforced with short glass fibers (GFRP)
- rubber matrix composites (RMC)
- metal matrix composites (MMC) such as titanium reinforced with carbon or ceramic fibers, short steel fiber reinforced concrete
- biomaterials

The objective of micromechanical modeling is to predict the interaction between the microstructure and the macroscopic (or overall or effective) properties.

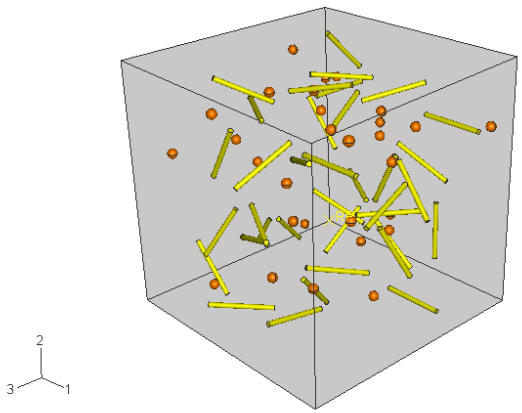


Figure 2-1 Matrix material reinforced with multiple phases of inclusions.

Micro-macro (two-scale) Approach

Consider a heterogeneous solid body whose microstructure consists of a matrix material and multiple phases of so-called *inclusions*, subjected to given loads and boundary conditions (BCs). The objective is to predict the influence of the microstructure on the response of the body. It would be computationally prohibitive to solve the mechanical problem at the scale of the microstructure.

Therefore, we distinguish two scales: the microscopic one (that of the heterogeneities) and the macroscopic one, where the solid can be seen as locally homogeneous. The link between the two scales is made via the concept of representative volume element (RVE).

At macro scale, each material point is supposed to be the center of a RVE, which should be sufficiently large to represent the underlying heterogeneous microstructure, and small with respect to the size of the solid body (see Figure 2-2). A two-scale approach which enables a transition between the two scales, both ways, is summarized in the following steps:

- **Micro → Micro Transition**

  1. Macro material point: center of a representative volume element (RVE)
  2. At micro scale: RVE contains a finite number of constituents
  3. Need a constitutive model for each of the constituents
  4. Micro/macro transition: homogenization method to find the macro constitutive response of RVE
  5. Continuum mechanics at macro scale with macro constitutive equation

- **Macro → Micro Transition**

  At each time and at each macro material point, do a numerical zoom in order to see what happens at the micro level (e.g., stresses and strains in each phase).

The following image shows an example of micro-macro transition.

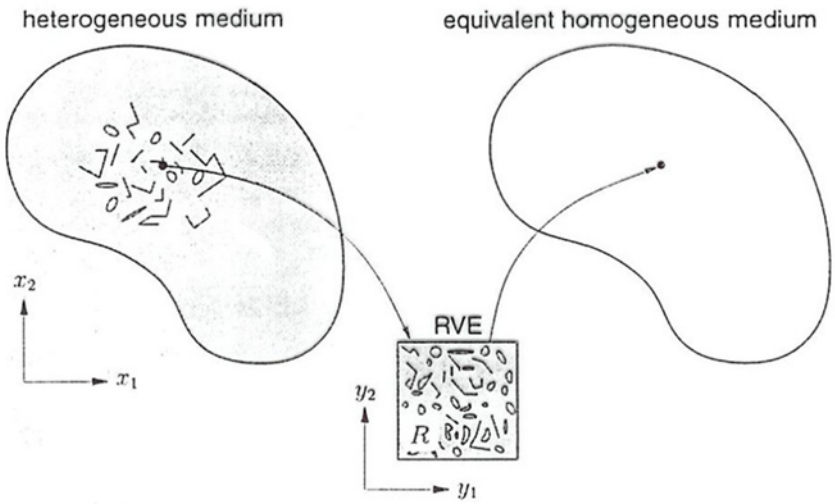


Figure 2-2 Micro-macro transition:- Upper left: microscopic scale, Upper right: macroscopic scale, Bottom: representative volume element (RVE). After Nemat-Nasser and Hori (1993).

General Averaging Results

The major difficulty in the two-scale (and more generally multi-scale) approach is to solve the RVE problems. In this section we give some very useful general averaging results. The notation and results are given hereafter. Classical continuum mechanics analysis is carried out at macro level. At each macro point **X**, we know the macro strain **E(X)** and need to compute the macro stress $\boldsymbol{\sigma}(\mathbf{X})$ or vice-versa.

The average quantity over a RVE (domain ω, volume V) is defined by,

$$\langle f(\mathbf{X}, \mathbf{x})\rangle = \frac{1}{V}\int_{\omega} f(\mathbf{X}, \mathbf{x})\,dV \tag{2-1}$$

where integration is performed with respect to micro coordinates, and $f(\mathbf{X}, \mathbf{x})$ is the micro field inside the RVE.

In the following, dependence on macro coordinates **X** will be omitted for simplicity. We consider two classical types of BCs: (1) linear displacements, and (2) uniform traction. The former corresponds to a given macro strain (or more accurately an imposed macro displacement gradient) and the latter to a known macro stress.

Macro Displacement Gradient G and Strain E = (G + G^T)/2

At micro level, the boundary $\partial\omega$ of the RVE is subjected to imposed linear displacements:

$$u_i(X) = G_{ij}x_j, \qquad X \in \partial\omega \tag{2-2}$$

Result: the average strain equals the macro strain, i.e., $\langle \varepsilon_{ij}\rangle = E_{ij}$

Macro Stress σ

At micro level, imposed traction on $\partial\omega$:

$$F_i(\mathbf{x}) = \sigma_{ij}n_j(\mathbf{x}), \qquad X \in \partial\omega \tag{2-3}$$

where **n** is the outward unit normal to $\partial\omega$.

Result: the average stress equals the macro stress i.e., $\langle \sigma_{ij}\rangle = \sigma_{ij}$ (2-4)

Conclusion

It appears that for a RVE under classical BCs, the macro strains and stresses are equal to the volume averages over the RVE of the unknown micro strain and stress fields inside the RVE. Another useful result is given hereafter:

Consider any self-equilibrated micro stress field and micro strain field

$$\sigma_{ij}^{*} = \sigma_{ji}^{*},\ \frac{\partial \sigma_{ij}^{*}}{\partial x_j} = 0,\ \forall x \in \omega \text{ and } \varepsilon_{ij}^{*} = \frac{1}{2}\left(\frac{\partial u_i^{*}}{\partial x_j} + \frac{\partial u_j^{*}}{\partial x_i}\right) \tag{2-5}$$

where u(**X**) is the micro displacement field associated with $\varepsilon^{*}(\mathbf{X})$.

Note: $\sigma^{*}(\mathbf{X})$ and $\varepsilon^{*}(\mathbf{X})$ are not necessarily related, and neither one is necessarily a solution to the micro problem.

If $\varepsilon^{*}(\mathbf{X})$ satisfies linear displacement B.C. (1) on $\partial\omega$ or $\sigma^{*}(\mathbf{X})$ satisfies uniform traction B.C. (2) on $\partial\omega$, then

$$\langle \sigma^{*} : \varepsilon^{*} \rangle = \langle \sigma^{*} \rangle : \langle \varepsilon^{*} \rangle \tag{2-6}$$

This is known as Hill's macro-homogeneity condition or Hill-Mandell condition, and is very useful for the derivation of homogenization models from variational formulations. In linear elasticity, the condition has a simple and powerful interpretation: if $\sigma^{*}(\mathbf{X})$ and $\varepsilon^{*}(\mathbf{X})$ are related, then the average of the micro energy equals the macro energy.

Homogenization and Scale-transition Methods

From a continuum mechanics viewpoint, the problem of homogenization can be stated as follows: at the macro scale, in each macro material point, if we know the macro strain, we need to compute the macro stress, and vice-versa. From the above results, the following fundamental conclusion was reached:

$$\text{Relating E and } \sigma \;\Leftrightarrow\; \text{Relating } \langle\varepsilon\rangle \text{ and } \langle\sigma\rangle$$

In linear elasticity, the problem can be stated in a simpler form: find the macro stiffness such that

$$\langle\sigma\rangle \;=\; \bar{\mathbf{C}} : \langle\varepsilon\rangle$$

This is illustrated in Figure 2-3.

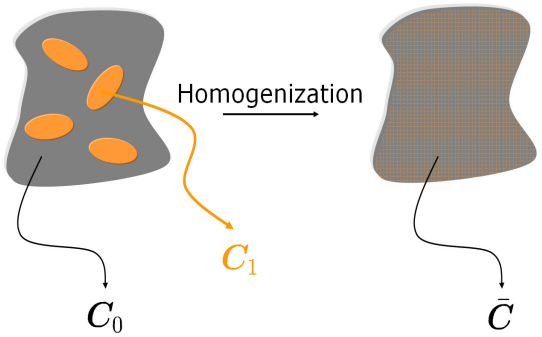


Figure 2-3 The idea of homogenization in linear elasticity. Left: heterogeneous material under given boundary conditions (BCs). Right: equivalent homogeneous material having the same effective stiffness under the same BCs.

This is the fundamental problem of homogenization in linear elasticity: find an equivalent homogeneous material which has the same effective macro stiffness as the real heterogeneous composite, under the same boundary conditions. There are a few scale-transition methods which address this problem: asymptotic or mathematical homogenization theory, method of cells, subcells and transformation field analysis, direct finite element analysis, and mean-field homogenization. The following two approaches are developed in the Digimat software suite:

Approach (1): Direct finite element analysis (FEA) of RVE at micro scale

This method is very general and accurate, and gives detailed micro fields. However, it has some serious drawbacks. For complex or simply realistic micro-structures, generating a good mesh can be very difficult and expensive in terms of user's time.

For nonlinear problems, the CPU time and memory usage can become prohibitive. Moreover, when FEA of a realistic structure at macro scale is coupled with FEA of a RVE in each macro point, then the direct FE approach becomes practically out of reach, especially in the nonlinear regime. This approach is the one followed in Digimat-FE.

Approach (2): Mean-field homogenization (MFH) models

This method is based on semi-analytical models and is the cornerstone of Digimat-MF. It will be developed in the remainder of this text, but its main advantages are: ease-of-use, speed (low CPU time) and reduced memory usage. Its main drawback is that it only gives approximations to the volume averages of stresses and strains, either at the macro level or in each phase.

Mean-field Homogenization (MFH)

The purpose of mean-field homogenization (MFH) is to compute approximate but accurate estimates of the volume averages of the stress and strain fields, both at the RVE level (macro stresses and strains) and in each phase. It is important to emphasize that MFH does not solve the RVE problem in detail, and therefore does not compute the detailed micro stress and strain fields in each phase.

There are different MFH models, each based on some specific assumptions. The simplest models are due to Voigt and Reuss. Voigt model assumes that the strain field is uniform inside the RVE. Consequently, the macro stiffness is found to be the volume average of the micro stiffnesses. In the Reuss model, the stress field is assumed to be uniform in the RVE. Therefore, the macro compliance (the inverse of the stiffness) is found to be the volume average of the micro compliances.

Voigt and Reuss models generalize the simple 1D models of bars in parallel, and in series, respectively. Both models are too simplistic. Indeed, assuming that strains or stresses are uniform within a composite is not realistic. Moreover, if each phase is isotropic, both models predict an isotropic composite, regardless of the shape and orientation of the inclusions, which is physically false. In the remainder of this text, we will study more sophisticated and satisfying MFH models. We first need to introduce some notation and results.

Two-phase Composites

Until otherwise indicated, we study simple two-phase composites made of a matrix material reinforced with a number of identical inclusions (I), having all the same material, shape and orientation. We use subscripts 0 for the matrix and 1 for the inclusions phase. The volume fractions in the two phases are such that $\nu_0 + \nu_1 = 1$.

The volume averages of the strain field over the RVE, the matrix phase and the inclusion phase are related as follows:

$$\langle \varepsilon \rangle_\omega = \nu_0 \langle \varepsilon \rangle_{\omega 0} + \nu_1 \langle \varepsilon \rangle_{\omega 1} \tag{2-7}$$

Actually, this identity holds for any micro field (e.g., stress field).

Any MFH model can be defined by so-called strain concentration tensors such that:

$$\langle \varepsilon \rangle_{\omega 1} = \mathbf{B}^\varepsilon : \langle \varepsilon \rangle_{\omega 0}, \quad \langle \varepsilon \rangle_{\omega 1} = \mathbf{A}^\varepsilon : \langle \varepsilon \rangle_\omega \tag{2-8}$$

The volume average of strain over all inclusions is related to the volume average of strain over the matrix phase via the first tensor, and to the volume average of strain over the entire RVE (macro strain) with the second tensor. The two strain concentration tensors are not independent. Indeed, the second one can be computed from the first one:

$$\mathbf{A}^\varepsilon = \mathbf{B}^\varepsilon : [\nu_1 \mathbf{B}^\varepsilon + (1 - \nu_1)\mathbf{I}]^{-1} \tag{2-9}$$

These results are valid for any material model for either phase.

Two-phase Linear Elastic Composites

We now focus on two-phase linear elastic composites, until otherwise indicated.

For any homogenization model defined by a strain concentration tensor, the macro stiffness (sub. 0 for matrix and 1 for inclusions) is:

$$\bar{C} = [\nu_1 C_1 : B^{\varepsilon} + (1-\nu_1) C_0] : [\nu_1 B^{\varepsilon} + (1-\nu_1) I]^{-1} \quad (2\text{-}10)$$

We have already seen the models of Voigt (uniform strain) and Reuss (uniform stress). They correspond to the following choices respectively:

$$B^{\varepsilon} = I \text{ and } B^{\varepsilon} = C_1^{-1} : C_0 \quad (2\text{-}11)$$

It can be shown that the real composite's stiffness is bounded by the Voigt and Reuss estimates, which provide upper and lower bounds, respectively. In practice however, they are far-apart bounds, and therefore of little use. Tighter bounds do exist and are due to Hashin and Shtrikman (1962).

Sophisticated MFH models or bounds closer than the Voigt and Reuss estimates all use a fundamental solution due to Eshelby (1957).

Eshelby's Problem

In a seminal paper published in 1957, J. D. Eshelby solved the following problem (see Figure 2-4).

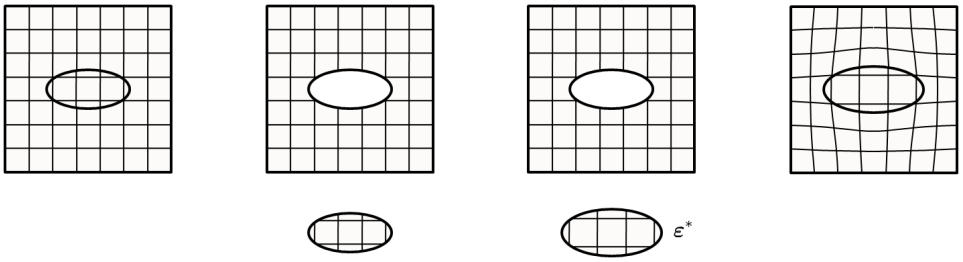


Figure 2-4 Eshelby's problem: an ellipsoidal volume within an infinite solid body of uniform stiffness is cut out, undergoes an eigenstrain and is welded back into the body.

Inside an infinite solid body of uniform stiffness $\mathbf{C_0}$, an ellipsoidal volume (I) is cut out, undergoes a stress-free eigenstrain ε^* and is then welded back into the cavity it occupied (see Figure 2-4).

Eshelby found that the strain inside the ellipsoidal volume (I) is uniform and related to the eigenstrain as follows:

$$\varepsilon(x) = \zeta(I, C_0):\varepsilon^* , \ \forall x \in (I) \tag{2-12}$$

where $\zeta(\mathbf{I}, \mathbf{C_0})$ is Eshelby's tensor.

It depends on $\mathbf{C_0}$ and the shape (not the size) and orientation of (I). If $\mathbf{C_0}$ is isotropic and ($\mathbf{I}$) is a spheroid (that is an ellipsoid of revolution), then the stiffness dependence is through Poisson's ratio only, and the shape dependence through the aspect ratio only. Eshelby's solution plays a fundamental role in MFH, as it enables to solve the single inclusion problem.

Single inclusion problem

An infinite solid body is subjected to linear displacements on its boundary corresponding to a uniform remote strain **E**. The body is made of a matrix phase of uniform stiffness $\mathbf{C_0}$ in which is embedded a single ellipsoidal inclusion (I) of uniform stiffness $\mathbf{C_1}$ (see Figure 2-5).

Using Eshelby's solution, this problem can be solved in closed form. It is found that the strain inside the inclusion ($\mathbf{I}$) is uniform and related to the remote strain as follows:

$$\varepsilon(x) = H^{\varepsilon}(I, C_0, C_1):E , \ \forall x \in (I) \tag{2-13}$$

where H^{ε} is the single inclusion strain concentration tensor, defined as follows:

$$H^{\varepsilon}(I, C_0, C_1) = \left\{ I + \zeta(I, C_0):C_0^{-1}:[C_1 - C_0] \right\}^{-1} \tag{2-14}$$

Another tensor which plays an important role is Hill's (polarization) tensor defined as:

$$P^{\varepsilon}(I, C_0) = \zeta(I, C_0):C_0^{-1} \tag{2-15}$$

The solution of the single inclusion problem is the cornerstone of well-known and successful MFH models.

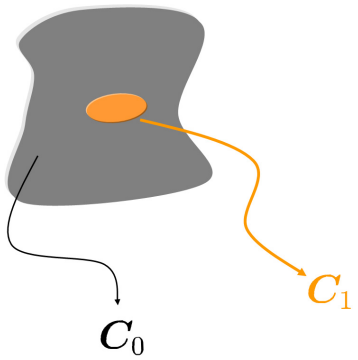


Figure 2-5 Single inclusion embedded in an infinite body.

MFH Models for Two-phase Composites

We now go back to the case of a composite RVE made of a matrix phase of uniform stiffness $\mathbf{C_0}$ reinforced with several inclusions of uniform stiffness $\mathbf{C_1}$, supposed to be so far identical in terms of material, shape and orientation. Linear displacements corresponding to a remote strain **E** are imposed on the boundary. Unlike the single inclusion problem, this multi-inclusion problem does not have an analytical solution. Therefore, several MFH models exist based on different assumptions. They all use the solution of the single inclusion problem.

Some examples follow hereafter.

Self-consistent (S-C) Model

The self-consistent (S-C) model supposes that each inclusion behaves as if it were isolated in a modified matrix which has the composite's unknown stiffness. The body is infinite and subjected to the remote strain **E**. Using the single inclusion solution, the strain inside each inclusion (I) of the real RVE is found to be:

$$\varepsilon(x) = H^{\varepsilon}(I, \bar{C}, C_1):E\,,\ \forall x \in (I) \tag{2-16}$$

The S-C model can be applied to general composite materials (not just simple two-phase composites) and even to materials without a matrix phase (polycrystals and aggregates). However, for real composites (having constituents with different material properties) the S-C model usually leads to bad predictions (e.g., too stiff).

This is the reason why S-C is not currently available in Digimat-MF.

Mori-Tanaka Model (1973)

This model was proposed by Mori and Tanaka (1973). The derivation is based on an approximate use of Eshelby's solution. It is found that the strain concentration tensor relating the volume average of strain over all inclusions to the mean matrix strain is given by:

$$B^{\varepsilon} = H^{\varepsilon}(I, C_0, C_1) \tag{2-17}$$

which is exactly the strain concentration tensor of the single inclusion problem. This led Benveniste (1987) to give the following simple interpretation of the Mori-Tanaka (M-T) model.

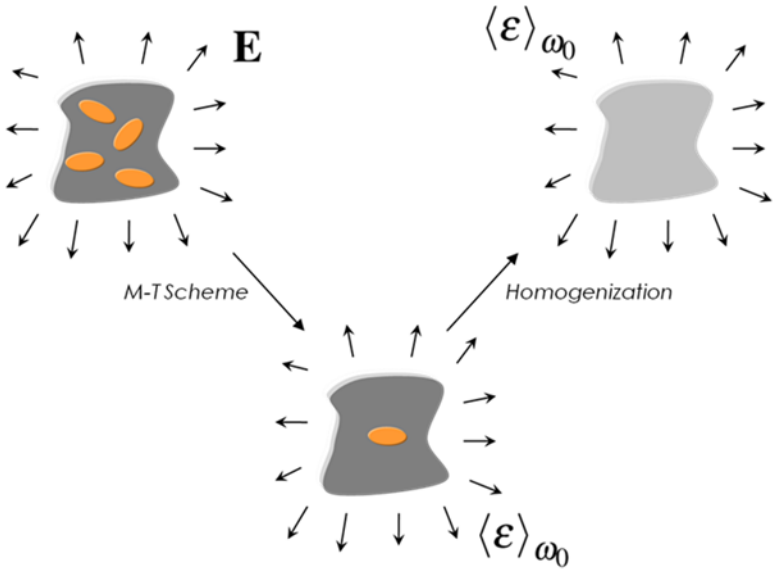


Figure 2-6 Illustration of the Mori-Tanaka (M-T) model.

Each inclusion in the real RVE behaves as if it were isolated in the real matrix. The body is infinite and subjected to the average matrix strains in the real RVE as the far field (remote) strain. This is illustrated in Figure 2-6.

The M-T model is very successful in predicting the effective properties of two-phase composites. In theory, it is restricted to moderate volume fractions of inclusions (less than 25% say) but in practice it can give good predictions well beyond this range.

Double Inclusion (D-I) Model

The Double inclusion (D-I) model was proposed by Nemat-Nasser and Hori (1993). It is based on the following ideas. Each inclusion ($\mathbf{I}$) of stiffness $\mathbf{C_1}$ is surrounded in its close surroundings with the real matrix material of stiffness $\mathbf{C_0}$, while outside those areas, there is a reference medium of stiffness $\mathbf{C_r}$. In other words, the real composite RVE is replaced with a model composite made

of a fictitious reference matrix of stiffness $\mathbf{C_r}$ in which are embedded inclusions of stiffness $\mathbf{C_1}$ coated with a material of stiffness $\mathbf{C_0}$ (hence the Double inclusion name; see Figure 2-7).

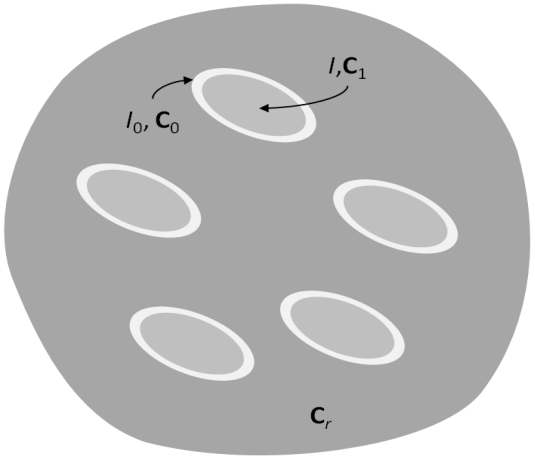


Figure 2-7 Illustration of the Double inclusion (D-I) model of Nemat-Nasser and Hori (1993).

The following inequality between the volumes and the volume fractions is supposed to hold:

$$\frac{V(I)}{V(I_0)} \geq \frac{\nu_1}{1-\nu_1} \tag{2-18}$$

Actually, the D-I model is a family of MFH models, as numerous schemes can be designed depending on the specific choice of the reference medium's stiffness. In particular, the following three cases can be demonstrated:

- $C_r = \bar{C}$ (composite): self- consistent model;
- $C_r = C_0$ (matrix): Mori-Tanaka model, $B^\varepsilon = H^\varepsilon(I, C_0, C_1) \equiv B_1^\varepsilon$
- $C_r = C_1$ (inclusion): inverse Mori-Tanaka model, $B^\varepsilon = [H^\varepsilon(I, C_1, C_0)]^{-1} \equiv B_u^\varepsilon$

The third case, the inverse M-T model can be retrieved directly from the real RVE with a permutation between the material properties of the inclusions and the matrix. This can be seen as the situation where the volume fraction of inclusions is so high that the inclusions almost form a continuous matrix phase. It was also demonstrated that M-T and inverse M-T estimates correspond to the Hashin-Shtrikman (H-S) bounds. Assuming that the inclusions are stiffer than the matrix, then M-T corresponds to the lower H-S bound, while inverse M-T gives the upper H-S bound.

The above results regarding M-T and inverse M-T have led Lielens (1999) to propose an interpolative D-I model defined by the following strain concentration tensor relating the mean strain over the inclusions to its counterpart over the matrix:

$$B^{\varepsilon} = [(1 - \xi(\nu_1))(B_1^{\varepsilon})^{-1} + \xi(\nu_1)(B_u^{\varepsilon})^{-1}]^{-1} \quad (2\text{-}19)$$

where $\zeta(\upsilon_1)$ is a smooth interpolation function chosen to be simply quadratic:

$$\xi(\nu_1) = \frac{1}{2}\nu_1(1 + \nu_1) \quad (2\text{-}20)$$

For linear elastic two-phase composites, the D-I model usually gives excellent predictions of effective properties, over all ranges of inclusions volume fractions, aspect ratios and stiffness contrasts (ratio of inclusions to matrix stiffness). This model is available in Digimat.

Distributed Orientations

So far, we have studied the case of a simple two-phase composite where all inclusions are identical and aligned. As a counter-example, consider a composite plastic part made of a thermoplastic polymer reinforced with short glass fibers. As typical of the injection molding manufacturing process, the fiber distribution inside the final product will vary widely in terms of orientation and length. Actually, inside each small volume (i.e., in each RVE) the fibers are misaligned (see Figure 2-8).

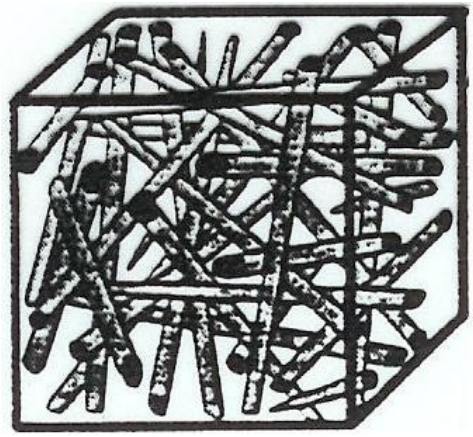


Figure 2-8 Misaligned short fibers inside a RVE, after Lielens (1999).

Before homogenizing such a composite material, we first need to introduce some tools in order to describe the fibers orientation. Actually, the following presentation is not restricted to fibers, it applies to spheroids (ellipsoids of revolution).

The orientation of each individual inclusion is described by a unit vector p along its axis of revolution, which in turn can be determined in 3D with two spherical angles θ and ϕ (see Figure 2-9).

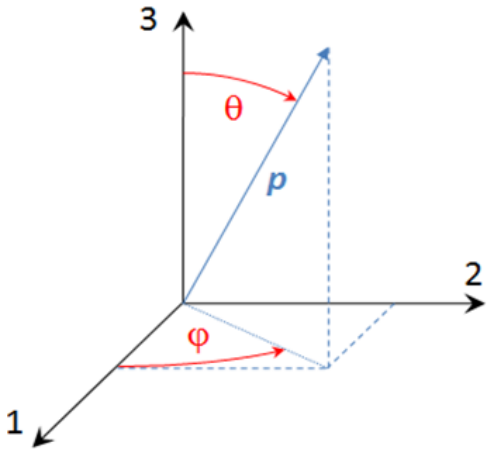


Figure 2-9 Orientation of a single inclusion.

Since the axis vector p varies from one inclusion to another within the same RVE, the notion of orientation distribution function (ODF) $\psi(p)$ is introduced. By definition $\psi(p)dp$ is the probability to find fibers within the solid angles [p, p + dp].

We now consider a rather general case where a matrix material is reinforced with N families of inclusions, each one defined by the same stiffness, aspect ratio and ODF:

- Matrix phase (domain ω_0): volume fraction v_0 and stiffness C_0;
- N inclusion families (i): v_i, C_i, AR_i, ODF $\psi_i(p)$.

Obviously, the volume fractions of matrix and inclusion families add up to 1:

$$v_0 + \sum_{i=1}^{N} v_i = 1 \tag{2-21}$$

Each ODF obeys two conditions:

$$\psi_i(p) = \psi_i(-p), \oint \psi_i(p)dp = 1 \tag{2-22}$$

The first equality simply means that two opposite axis vectors define indistinguishable inclusions, and the second identity is a normalization condition imposing that the sum of probabilities equals to 1.

The homogenization of such composites in Digimat-MF is carried out in two steps which are illustrated in Figure 2-10. The real composite RVE (the top structure in the figure) is replaced

with a model RVE which is an aggregate of so-called pseudograins (the middle-left structure in the figure).

Each pseudo-grain occupies a domain: $\omega_{i,\,p}$ and is a basic two-phase composite made of a matrix phase (in concentration ν_0) reinforced with identical and aligned inclusions from family (i), of orientation between p and p+dp.

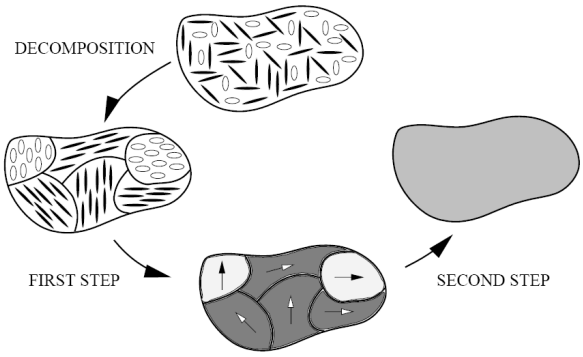


Figure 2-10 Two-step homogenization procedure for a composite with misaligned inclusions. Top: real RVE. Middle left: decomposition into an aggregate of pseudo-grains. Bottom: homogenization of each pseudo-grain (first step). Middle right: homogenization of the aggregate of homogenized pseudo-grains (second step). After Lielens (1999).

The homogenization of the model RVE is performed in two steps.

1. Each pseudograin is homogenized using a MFH model appropriate for basic two-phase composites (e.g., Mori-Tanaka or interpolative Double inclusion).
2. The effective response of the set of homogenized pseudo-grains is computed.

In the current version of Digimat-MF, the Voigt model is used in this second step. Recall that Voigt is inappropriate for a real composite, but for an aggregate (step 2 in this procedure) our experience shows that it gives good and physically acceptable predictions, especially in the most common case of N = 1 (that is one family of inclusions)

Orientation Tensors

In the misaligned orientation case, we can homogenize the composite RVE if we know the ODF. However, there are two problems:

- The ODF is not known in practice.

  For instance, in a real part made of a thermoplastic polymer reinforced with short glass fibers, and manufactured with the injection molding process, the ODF (actually a field of ODFs) is usually not measured experimentally and not predicted numerically.

  Instead, available injection molding software predict a so-called second-rank orientation tensor.

- There are some important cases where the ODF itself is not needed, and the composite can be homogenized based on the knowledge of the second- and fourth-rank orientation tensors only.

Before defining orientation tensors, we need to introduce the notion of orientation averaging. Let $\mu(\mathbf{p})$ be an orientation-dependent field inside the RVE. Its orientation average is the integral over all orientations, weighted by the ODF:

$$\langle \mu(p) \rangle_{\psi_i} = \oint \mu(p)\psi_i(p)dp \tag{2-23}$$

The second- and fourth-rank orientation tensors **a** and **A** are the following orientation averages:

$$a \equiv \langle p \otimes p \rangle_{\psi}, \; A \equiv \langle p \otimes p \otimes p \otimes p \rangle_{\psi} \tag{2-24}$$

Tensors **a** and **A** are statistical second- and fourth- order moments of the orientation field p, respectively. Both tensors give information about the average orientation of inclusions inside the RVE. From their definition, it is seen that these tensors have to satisfy a number of conditions:

$$a_{ij} = a_{ji}, \; a_{11}, a_{22}, a_{23} \geq 0, \; a_{ii} = 1 \tag{2-25}$$

$$A_{ijij} \geq 0 \quad (\text{no sums}), \; A_{ijll} = a_{ij} \tag{2-26}$$

Digimat-MF checks the user's input regarding second-rank tensor **a**. The latter is stored as a 3 x 3 symmetric matrix. Some special expressions of the orientation tensor are:

1. diag(1, 0, 0) for aligned orientation in direction 1
2. diag(1/2, 1/2, 0) for random orientation in the plane orthogonal to direction 3
3. diag(1/3, 1/3, 1/3) for random orientation in 3D

Closure Approximations

In practice, for a composite with misaligned inclusions, usually only the second-rank orientation tensor **a** is given, not the ODF. This is typically the case for short glass fiber reinforced thermoplastics, where fiber orientation is predicted using an injection molding software. Recall that the ODF itself is usually unknown.

There are cases however where both the second- and fourth-rank orientation tensors are needed. An important example is a linear elastic composite made of isotropic phases where homogenization needs only the second and fourth-rank orientation tensors.

The issue is then to compute the fourth-rank orientation tensor **A** from the only knowledge of the second-rank tensor **a**. This is called the *closure* problem. For random orientation, an exact solution exists and is called the linear closure:

$$A^{l}(a) = \alpha_l 1 \otimes 1 + 2(\alpha_l - \beta_l)I^s + \beta_l(1 \otimes a + a \otimes 1) + 2\beta_l[I(a + 1) - I(a)] \tag{2-27}$$

$$\alpha_1 = -\frac{1}{35} \text{ and } \beta_1 = \frac{1}{7} \text{ in 3D, } \alpha_1 = -\frac{1}{24} \text{ and } \beta_1 = \frac{1}{6} \text{ in 2D} \quad (2\text{-}28)$$

For aligned orientation, an analytical solution exists and is the quadratic closure:

$$A^q(a) = a \otimes a \quad (2\text{-}29)$$

For all other orientations, i.e., those which are neither aligned nor random, there exists no closed-form solutions, only closure approximations. The implemented method in Digimat is the orthotropic closure of Cintra and Tucker (1995).

Remark (ODF reconstruction): We have already emphasized the fact that the ODF is usually unavailable. However, there are cases where it is needed for homogenization. Examples are: inelastic composite materials, or linear elastic composites with an anisotropic matrix. One procedure used in Digimat is to approximate the fourth-rank orientation tensor **A** from the second-rank tensor **a**, and then to reconstruct the ODF approximately using the theory proposed by Onate and Leckie (1998).

Multi-phase Composites

Multi-phase composites are made of a matrix material and at least two inclusion phases which differ in terms of materials, aspect ratio or orientation. The two-step homogenization method which was presented earlier can be used, as it is very general and not restricted to misaligned inclusions described by an ODF.

The method is depicted in Figure 2-11 and its main steps are:

1. Decomposition of the RVE into a set of pseudograins, each being a basic two-phase composite with identical and aligned inclusions
2. Homogenization of each pseudo-grain (e.g., with Mori-Tanaka model)
3. Homogenization of the aggregate of homogenized pseudo-grains (with Voigt scheme)

However, there are some multi-phase composites for which the two-step method can lead to bad or even physically unacceptable predictions. This is the reason why another homogenization procedure, the multi-level method, is also available in Digimat-MF. The method is based on nested homogenization levels. It is illustrated in Figure 2-11 for a three-phase composite and proceeds as follows:

At the deepest level (2), the real matrix material is homogenized with a first family of (black) inclusions. The effective material thus obtained plays the role of a fictitious (grey) matrix which is reinforced with another set of (red) inclusions to constitute an upper level (1) composite. This procedure is repeated until all inclusion families have been accounted for. Usually, the Mori-Tanaka model is used at a given level, if the composite at that level corresponds to a basic two-phase composite. If at one level, the inclusions are misaligned and described by an ODF, then at that level a two-step Mori-Tanaka/Voigt homogenization scheme is used.

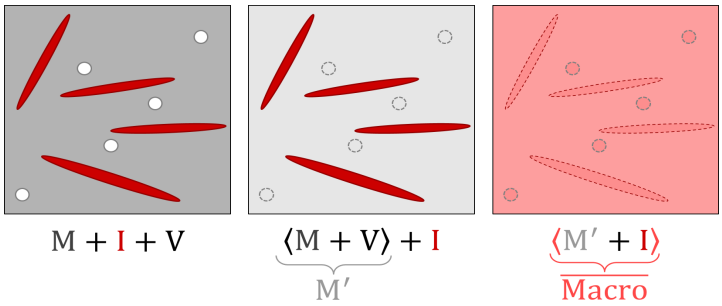


Figure 2-11 Multi-level method. Left: three-phase composite to be homogenized. Middle: corresponding two-phase composites at "higher level" where fictitious matrix (in light grey) is obtained after homogenization at "deepest level" between matrix (dark grey) and voids (white). Right: homogenized materials obtained at higher level.

The multi-level method is capable of delivering excellent predictions when the right choice of the nested homogenization levels is made. Indeed, the choice of the inclusion family to be used at each level has an influence, sometimes a strong one, on the final predictions. It is difficult to propose recommendations as more experience is needed. However, it seems that for inclusions with different material properties, when going from the deepest level to the upper level, the inclusions should be added from the most compliant to the most rigid.

While the two-step method is very general and available for all material models and all combinations, the current implementation of the multi-level method is more restricted (see section Multi-level Scheme).

Remark: composites with coated inclusions

The multi-level method was also developed for composites with coated inclusions, but with a particular choice of homogenization levels. Indeed in this case, in the deeper level, the inclusions are homogenized with their coatings leading to "equivalent" inclusions. The latter are homogenized with the real matrix in the upper level.

This is illustrated in Figure 2-12. For comparison purposes, the two-step method is also depicted in the same figure. It leads to different pseudo-grains: real matrix with the inclusions on the one hand, and with the coatings on the other. In the two-step method, the coatings are considered as if they were isolated inclusions, which is physically incorrect.

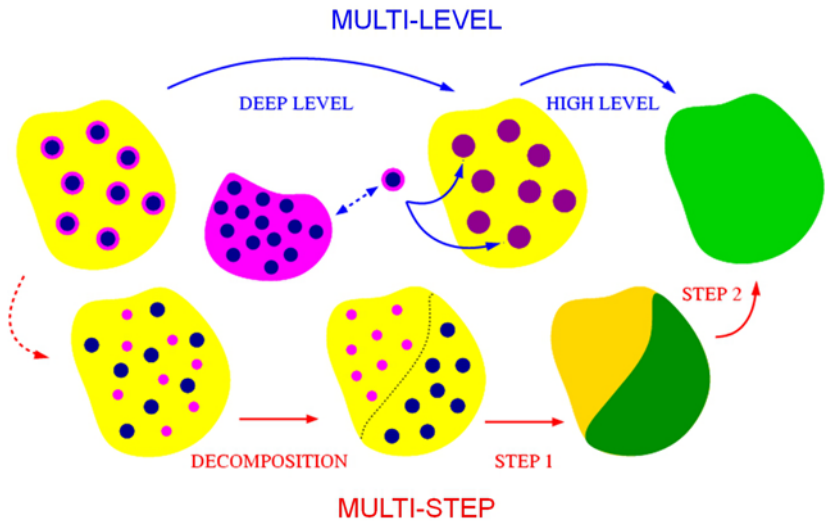


Figure 2-12 Composite with coated inclusions. Illustration of the multi-level method (top figures) and the two-step method (bottom figures).

Linear Thermo-elastic Composites

So far, we have studied the isothermal case, we now consider linear thermoelastic composites. A basic two-phase composite is made of a matrix (subscript 0) and identical and aligned inclusions (subscript 1). More complex composites (distributed orientations, multiple phases, coatings) can be studied with the appropriate methods which were presented in the isothermal case.

Each homogeneous material phase obeys the following constitutive equations:

$$\begin{aligned}\sigma_0(x) &= C_0(\varepsilon_0(x) - \alpha_0\Delta T) & \sigma_1(x) &= C_1(\varepsilon_1(x) - \alpha_1\Delta T)\\ &= C_0\varepsilon_0(x) + \beta_0\Delta T & &= C_1\varepsilon_1(x) + \beta_1\Delta T\end{aligned} \tag{2-30}$$

The elastic stiffness and the coefficients of thermal expansion (CTEs) are respectively denoted by **C** and α_{ij},

while $\beta = -C:\alpha$

The composite RVE is subjected to linear displacements on its boundary corresponding to a macroscopic strain, and to a uniform change in temperature.

Objective: find the corresponding composite elastic stiffness and thermal expansion tensors so that:

$$\begin{aligned} \langle \sigma \rangle &= \bar{C}:(E - \bar{\alpha}\Delta T) \\ &= \bar{C}:E + \beta\Delta T \end{aligned} \tag{2-31}$$

In the isothermal case, MFH models for basic two-phase composites (e.g., Mori-Tanaka or Double inclusion) are defined by their strain concentration tensors $\mathbf{B}^{\varepsilon}$ (or $\mathbf{A}^{\varepsilon}$). In thermo-elasticity, it is found that the mean strain over all inclusions is related to the macroscopic strain as follows:

$$\langle \varepsilon \rangle_{\omega 1} = A^{\varepsilon}:E + a^{\varepsilon}\Delta T \tag{2-32}$$

with $\mathbf{A}^{\varepsilon}$ identical to the linear elastic case, with the same homogenization scheme and

$$\begin{aligned} a^{\varepsilon} &\equiv (A^{\varepsilon} - I):(C_1 - C_0)^{-1}:(\beta_1 - \beta_0) \\ E &= \nu_0 \langle \varepsilon \rangle_{\omega 0} + \nu_1 \langle \varepsilon \rangle_{\omega 1} \end{aligned} \tag{2-33}$$

The composite's macro response can then be predicted:

$$\langle \sigma \rangle = \bar{C}:E + \beta\Delta T \tag{2-34}$$

with the elastic stiffness tensor identical to the isothermal case,

$$\bar{C} = [\nu_1 C_1:B^{\varepsilon} + (1 - \nu_1)C_0]:[\nu_1 B^{\varepsilon} + (1 - \nu_1)I]^{-1} \tag{2-35}$$

and

$$\bar{\beta} = \nu_0\beta_0 + \nu_1\beta_1 + \nu_1(C_1 - C_0):a^{\varepsilon}, \qquad \bar{\alpha} = -\bar{C}^{-1}:\bar{\beta} \tag{2-36}$$

Note that the macro CTE is not an independent property, it depends on the stiffness of each phase and also on the macro stiffness.

Linear Viscoelastic Composites

Some materials such as polymers exhibit a viscoelastic behavior at small strains. This is characterized by the following so-called viscous effects:

- creep (increase of strain at constant stress)
- relaxation (decrease of stress at constant strain)
- strain rate dependence (increase of stiffness with strain rate)
- dissipation of energy in a closed loading cycle.

In a tension test with loading and unloading, the loading and unloading paths are different, although a zero strain is recovered at zero stress if we wait long enough. Contrary to elasticity theory, a viscoelastic model is able to simulate those phenomena. At very large (infinite) times, the viscoelastic response tends towards the elastic one.

In viscoelasticity, the stress at a time t depends on all the strain history up to that time (memory effect) as translated by the first integral equation hereafter

$$\sigma(t) = G(t):\varepsilon(0) + \int_0^t G(t-\tau):\frac{\partial \varepsilon(\tau)}{\partial \tau} d\tau \tag{2-37}$$

G designates the fourth-rank tensor of relaxation moduli. It is time dependent and, in the isotropic case, it is defined by two time-dependent moduli, the bulk and the shear moduli. These usually obey a *Prony* series (see Viscoelasticity). In linear viscoelasticity, the tensor **G** (or bulk and shear moduli) depends on time, but not on the strain itself. Therefore, multiplying all the strain history by a constant factor, multiplies the stress response by the same factor. A similar result holds for summing up strain histories. All this explains the difference between linear and nonlinear viscoelasticity.

Using the Laplace-Carson transform (L-C),

$$\hat{f}(s) = s\int_0^\infty f(t)e^{-st}dt \tag{2-38}$$

the convolution product can be reduced to a simple multiplication and the viscoelastic constitutive equation brought to a form similar to that of linear elasticity

$$\hat{\sigma}(s) = \hat{G}(s):\hat{\varepsilon}(s) \leftrightarrow \sigma = C:\varepsilon \tag{2-39}$$

As the transformed viscoelastic constitutive equation is form-identical to that of linear elasticity, all MFH models and procedures available for linear elasticity can be used for linear viscoelasticity, but in the L-C domain. The important practical problem is to inverse transform these values, from the L-C domain to the time one, so that they can be interpreted and used easily. In Digimat, this is accomplished via numerical inversion of the L-C transforms, using the method of Schapery (1962) which is summarized below.

Assume the L-C transform of an unknown time function f (t) can be evaluated at any point in the transformed domain. The approximation to the time domain function can be developed into a n-terms Dirichlet series with an additional affine term

$$\tilde{f}(t) = A + Bt + \sum_{k=1}^{n} b_k \underbrace{\left(1 - e^{-t/\theta_k}\right)}_{\text{basis functions}} \tag{2-40}$$

whose transform is then fitted with the exact transform of the unknown time function at a number **m** of so-called collocation points

$$\hat{f}(s_l) = A + \frac{B}{s_l} + \sum_{k=1}^{n} \frac{b_k}{1 + s_l \theta_k}, \qquad 1 \le l \le m \tag{2-41}$$

The number of collocation points and their location should be chosen carefully in order to achieve the right compromise between accuracy, CPU time and memory space.

Rate-independent Inelastic Composites

In this section, we consider a class of nonlinear inelastic material models whose response is unaffected by the strain rate (or loading rate). These are typically elasto-plastic models (e.g., J_2-plasticity with isotropic hardening, J_2-plasticity with Chaboche's nonlinear kinematic hardening, pressure-dependent Drucker Prager), but also rate-independent damage models (e.g., J_2-plasticity with isotropic hardening coupled with Lemaitre- Chaboche ductile damage model).

For nonlinear material models, a closed-form solution for the single inclusion problem is unavailable. Several steps need to be taken in order to generalize, in an approximate manner, Eshelby's solution and useful mean-field homogenization (MFH) schemes (such as Mori-Tanaka) from linear elasticity to the nonlinear regime. The main issues are: linearization, comparison materials, first- versus second-order homogenization and isotropization.

Let us start with the issue of linearization. Three methods for a uniaxial stress state are illustrated in Figure 2-13. The incremental method consists in taking the tangent to the stress-strain curve and writing a pseudo-linear relation (y = a.x) where y and x are increments or rates of stress and strain, respectively, and a the slope of the tangent to the stress-strain curve. The affine linearization method also takes the tangent but writes a pseudo-affine relation (y = a.x+b), where y is the stress, x: the strain, a: the slope of the tangent and b: a fictitious stress at zero strain. Finally, the secant formulation assumes a pseudo-linear relation (y = a.x) where y is the stress, x: the strain, and a: the secant stiffness.

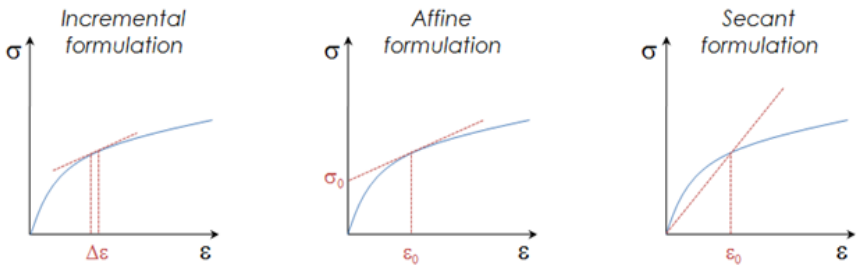


Figure 2-13 Illustration of three linearization methods: incremental, affine and secant.

The latter method is a total deformation theory, similar to nonlinear elasticity, and is restricted to monotonic and proportional loadings. The incremental and affine methods are much more

general, and are able to deal with any loading history, through appropriate time discretization schemes. However, since the affine method is mainly useful for rate-dependent inelastic models (e.g., elasto-viscoplasticity), we will focus in this section on the incremental formulation.

Consider a two-phase composite material made of rate-independent inelastic phases, and subjected to linear displacements on its boundary, which evolve with time (t), or a time-like parameter. Using the incremental formulation, the stress and strain rates in each material point of each phase (r = 0, 1) are related through a tangent operator as follows:

$$\dot{\sigma}(x, t) = C_r(\varepsilon(x, t), t):\dot{\varepsilon}(x, t), \qquad \forall x \in \omega_r \qquad r = 0, 1 \tag{2-42}$$

This relation looks like linear elasticity where the stress and strain are replaced with stress and strain rates, and the tangent operator takes the place of Hooke's elastic stiffness tensor. Therefore, known homogenization schemes could be applied. There is an important difference however between linear elasticity and elastoplasticity or rate-independent inelasticity in general. Indeed, in the nonlinear regime, since the stress and strain fields are not uniform in each phase, then the tangent operator is not uniform per phase either, while in linear elasticity, Hooke's operator is uniform per phase.

The heterogeneity of the tangent operator within each phase makes impossible the generalization of Eshelby's result and useful MFH schemes such as Mori-Tanaka. A workaround consists in defining fictitious comparison materials in each phase such that by definition, they possess a tangent operator which is uniform within the phase (but varies with time or a time-like parameter).

$$\dot{\sigma}(x, t) = \hat{C}_r(t):\dot{\varepsilon}(x, t), \qquad \forall x \in \omega_r \qquad r = 0, 1 \tag{2-43}$$

The concept of comparison materials is illustrated in Figure 2-14.

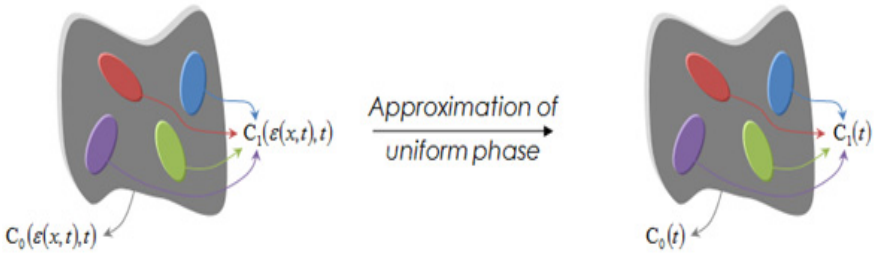


Figure 2-14 The concept of comparison materials for composites made of inelastic materials Left: real tangent operators are heterogeneous within each phase. The right part of the figure shows the fictitious comparison materials have uniform tangent operators within each phase.

Using the notion of linearization together with that of comparison materials enables to generalize Eshelby's results and MFH models. Now the remaining issue is the practical definition or computation of the tangent operators of comparison materials.

There are two methods:

- **The first-order homogenization method:** In this method, the comparison material of each phase is computed by using the real material model of the phase with the volume average of the strain field in the phase. The computed tangent operator is taken as the uniform comparison tangent, and the computed uniform stress is an approximation of the volume average of the stress field in the phase. This is depicted in Figure 2-15.

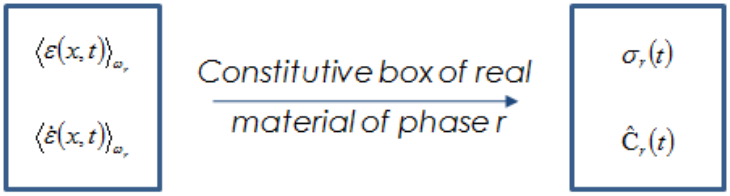


Figure 2-15 Illustration of the concept of first-order homogenization.

The material model of the phase is called with the volume average of the strain and strain increment (or rate) fields in the phase. A volume average of a field is called first statistical moment, hence the name 'first-order' homogenization.

- **Second-order homogenization method:** In this method, the second statistical moments of strain fields are also used in order to define comparison materials.

Isotropization of Comparison Tangent Operators

Consider the common case of an elasto-plastic matrix reinforced with stiffer elastic inclusions. If we homogenize this kind of composites, we find the typical results reported in Figure 2-16.

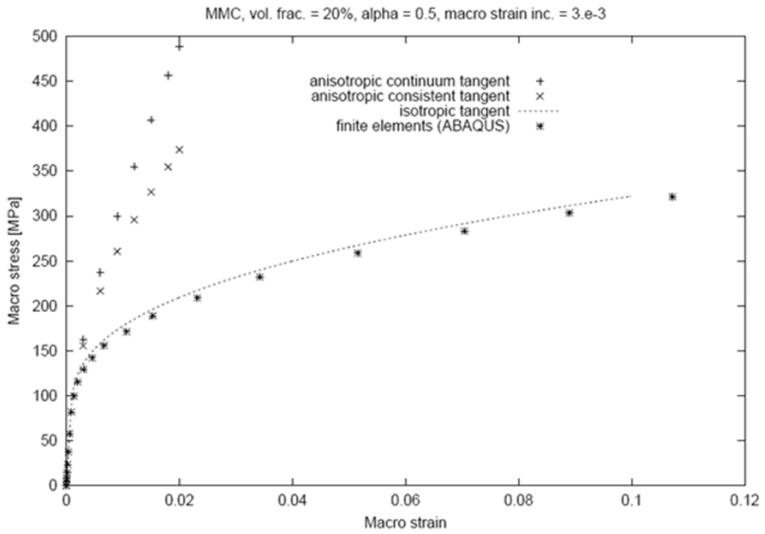


Figure 2-16 Prediction of the macro stress-strain response of a metal matrix composite.

The figure shows prediction of the macro stress-strain response of a metal matrix composite using different computations of the macro tangent operator and comparison against reference FE results (Doghri and Ouaar, (2003)).You will find responses (plus and cross signs in the figure) which are much stiffer than the reference FE results.

The problem and the possible solutions to it are too complex to explain in detail. Nevertheless, a few conclusions from research work can be given. It appears that one of the key issues is that the incremental formulation is based on a matrix comparison tangent operator which is anisotropic, and that much better predictions can be obtained if an isotropic part of that tangent operator is used appropriately. This is illustrated in the above figure where MFH predictions (dotted line) agree extremely well with the target FE results (star signs).

Some Notation

- **1**: 2nd-order identity tensor, $1_{ij} = \delta_{ij}$, where δ_{ij} is the Kronecker operator, i.e., $\delta_{ij} = 1$ if i = j and 0 otherwise.
- **I**: the fourth-rank symmetric identity tensor, $I_{ijkl} = \frac{1}{2}(\delta_{ik}\delta_{jl} + \delta_{il}\delta_{jk})$.
- $\mathbf{I^{vol}}$: the spherical operator, $\mathbf{I^{vol}} \equiv \mathbf{1} \otimes \mathbf{1}$, $\mathbf{I^{vol}} : \sigma = \sigma_{mm}/3\ \mathbf{1}$.
- $\mathbf{I^{dev}}$: the deviatoric operator, $\mathbf{I^{dev}} \equiv \mathbf{I} - \mathbf{I^{vol}}$, $\mathbf{I^{dev}} : \sigma = \mathrm{dev}(\sigma)$.
- $\mathbf{C^{el}}$: Hooke's elasticity operator. In the isotropic case, it is given by $\mathbf{C^{el}} = 3K\mathbf{I^{vol}} + 2G\mathbf{I^{dev}}$.

Tangent Operators for the J_2-plasticity Model

For elasto-plasticity with $\mathbf{J_2}$ flow theory and isotropic hardening, a continuum tangent operator relating stress and strain rates has the following expression:

$$C^{ep} = C^{el} - \frac{(2G)^2}{h}(N \otimes N), \qquad N = \frac{\partial f}{\partial \sigma} = \frac{3}{2}\frac{\mathrm{dev}(\sigma)}{\sigma_{eq}}, \qquad h = 3G + \frac{dR}{dp} \tag{2-44}$$

We designate by **p** the accumulated plastic strain, **R(p)** the hardening stress and by **N** the normal to the yield surface in stress space. It is seen that although the material model is isotropic, the tangent operator is anisotropic, because of the presence of the dyadic term in **N**.

On the other hand, numerically, the rate equations are discretized in time using a fully implicit backward Euler scheme. After convergence of the return mapping algorithm, the discrete equations are linearized w.r.t. all variables at the end of the time step. An algorithmic (or consistent) tangent operator is then defined by relating the total variations of stress and strain at the end of the time step,

$$C^{alg} = C^{ep} - (2G)^2(\Delta p)\frac{\sigma_{eq}}{\sigma_{eq}^{tr}}\frac{\partial N}{\partial \sigma}, \qquad \frac{\partial N}{\partial \sigma} = \frac{1}{\sigma_{eq}}\left(\frac{3}{2}I^{dev} - (N \otimes N)\right) \tag{2-45}$$

Again, it is seen that this tangent operator is anisotropic, despite the isotropy of the material model. Designating by $\Delta \mathbf{p}$ the plastic multiplier increment over a time step, it is found that,

$$\mathbf{C}^{alg} \rightarrow \mathbf{C}^{ep}, \text{ if } \Delta p \rightarrow 0 \tag{2-46}$$

Two isotropization methods are implemented in Digimat-MF: the spectral and the general methods.

Spectral Isotropization Method: Application to the J_2-plasticity Model

The spectral isotropization method was proposed by Ponte-Castañeda and applies to tangent operators which can be written under the following form

$$\mathbf{C}^{ani} = 3k_1\mathbf{C}^{(1)} + 2k_2\mathbf{C}^{(2)} + 2k_3\mathbf{C}^{(3)} \tag{2-47}$$

where

$$\mathbf{C}^{(1)} = \mathbf{I}^{vol}, \qquad \mathbf{C}^{(3)} = \frac{2}{3}\mathbf{N} \otimes \mathbf{N}, \qquad \mathbf{C}^{(2)} = \mathbf{I}^{dev} - \mathbf{C}^{(3)} \tag{2-48}$$

These tensors satisfy the following conditions:

$$\mathbf{C}^{(1)} + \mathbf{C}^{(2)} + \mathbf{C}^{(3)} = \mathbf{I}, \qquad \mathbf{C}^{(i)}:\mathbf{C}^{(j)} = \delta_{ij}\mathbf{C}^{(i)} \text{ (no sum over i)} \tag{2-49}$$

It appears that for the J_2-plasticity model, both the continuum and algorithmic tangent operators can be written under the spectral decomposition format. The isotropic projection is defined by two tangent moduli: in bulk and shear. After some algebra, their expressions are found to be, e.g., Doghri and Ouaar (2003),

$$K_t = K_e \tag{2-50}$$

$$G_t = G_e\left(1 - \frac{3G_e}{h}\right) \tag{2-51}$$

These expressions are common to both tangent operators, the continuum and the algorithmic. Physically, it was found that the spectral decomposition method corresponds to a stiffness reduction of the continuum tangent in a direction orthogonal to N (Chaboche et al., (2005); Pierard and Doghri, (2006a)).

General Isotropization Method: Application to the J_2-plasticity Model

The general isotropization method applies to any anisotropic tangent operator. First, when the following notation is used for fourth-rank tensors **A** and **B**:

$$\mathbf{A}::\mathbf{B} \equiv A_{ijkl}B_{lkji} \tag{2-52}$$

one can check that any symmetric and isotropic fourth-rank tensor $\mathbf{C}^{iso}$ can be written as follows:

$$C^{iso} - (I^{vol}::C^{iso})I^{vol} + \frac{1}{5}(I^{dev}::C^{iso})I^{dev} \tag{2-53}$$

The general method uses a similar definition in order to define the isotropic projection of any anisotropic fourth-rank tensor C^{ani} as follows:

$$C^{iso} = (I^{vol}::C^{ani})I^{vol} + \frac{1}{5}(I^{dev}::C^{ani})I^{dev} \tag{2-54}$$

When applied to the J_2-plasticity model, the general isotropization method leads to the following expressions for tangent bulk and shear moduli,

$$K_{tan} = K, \qquad G_{tan} = G - \frac{3}{5}G^2\left[\frac{1}{h} + 4\frac{\Delta p}{\sigma_{eq}^{tr}}\right] \tag{2-55}$$

The tangent shear modulus is different from the one obtained with the spectral isotropization method.

Isotropization Methods: Recommendations

The issue of isotropization was extensively studied by Pierard and Doghri, (2006a), who made the following recommendations which are followed in Digimat-MF.

When the spectral decomposition applies, e.g., J_2-plasticity model, then use it and compute Hill's tensor with the spectral isotropic projection of the matrix comparison tangent operator. The rest of the computations are performed with each phase's anisotropic comparison tangent operator.

However, when the spectral method does not apply, e.g., algorithmic tangent operator of Chaboche's model with combined nonlinear isotropic and kinematic hardenings, or Lemaitre-Chaboche ductile damage model, or Drucker-Prager pressure-dependent model, then the general isotropization method is used and Eshelby's tensor should be computed with it. Again, all other computations are carried out with each phase's anisotropic comparison tangent operator.

We recall that Eshelby's tensor, denoted by $\varsigma(\mathbf{I}, \mathbf{C_0})$ depends on the geometry of the inclusions, i.e., their orientation and aspect ratio, and on the matrix comparison tangent modulus $\mathbf{C_0}$ during the time increment.

Using Eshelby's tensor, Hill's tensor or the polarization tensor can be introduced

$$P(I, C_0) \equiv \zeta:(C_0)^{-1} \tag{2-56}$$

Numerous results show that the above recommendations lead to good MFH predictions at macro level, and often at the micro level as well. However, there are cases where the macro predictions are well off reference results, either FE or experimental ones. Typical examples are short fiber reinforced thermoplastic polymers.

In those cases, a second-order MFH usually leads to significant improvements over the first-order predictions. However, for glass fiber thermoplastics, the improvement is usually insufficient to close the gap with reference results. At present time, there is still no fully predictive theoretical solution for this difficult problem. In Digimat-MF, a practical solution is proposed, called the *modified spectral method*. The idea is to modify the expression of the tangent shear modulus given by the spectral isotropization method in a heuristic manner.

The method introduces parameters which can be fixed by fitting against results on the composite level, obtained either experimentally, or with Digimat-FE. More details on the modified spectral isotropization method can be found in Isotropic Extraction Methods section.

Rate-dependent Inelastic Composites

In this section, we consider a class of nonlinear inelastic material models whose response changes with the strain rate (or the loading rate). In Digimat-MF, two such models are currently available.

- The first one is J_2- viscoplasticity, (EVP) which extends the classical J_2-plasticity model to the rate-dependent viscoplastic regime.
- The other one is a fully coupled viscoelastic-viscoplastic model (VE-VP) which couples linear viscoelasticity with J2-viscoplasticity.

In other words, the linear elastic response in the EVP model is replaced by a linear viscoelastic response in the VE-VP model. More details about these material models are given in their respective related sections.

Rigorously, the incremental formulation should not be used in the present case because a continuum tangent operator C^{in} relating stress and strain rates does not exist in rate-dependent inelasticity, i.e.,

$$\dot{\sigma}^{in}(t) \neq C^{in}(t):\dot{\varepsilon}^{in}(t) \tag{2-57}$$

An appropriate theory, called the discrete affine linearization method, was developed by Doghri et al. (2010) and is implemented in Digimat-MF. Accordingly, the stress and strain increments are related by the following relations

$$\Delta\sigma = C^{alg}(t_{n+1}):(\Delta\varepsilon - \Delta\varepsilon^{af}) \tag{2-58}$$

where $C^{alg}(t_n+1)$ is exactly the algorithmic tangent operator at time t_n+1, defined as follows

$$C^{alg}(t_{n+1}) = \frac{\partial\sigma}{\partial\varepsilon}(t_{n+1}) \tag{2-59}$$

$\Delta\varepsilon^{af}$ is a term called the affine strain increment.

For the J_2-viscoplasticity model, the complete expressions are given in Doghri et al. (2010).

For the coupled VE-VP model, the discrete affine linearization method was extended by Miled et al. (2011) and leads to a relation between stress and strain increments which is form-similar to the above relation in EVP.

For both EVP and VE-VP models, since the incremental stress/strain relation has the same format as linear thermo-elasticity, then linear thermo-elastic MFH models can be used, for each time step and each iteration.

The other issues are: comparison materials, first-order homogenization, and isotropization of anisotropic comparison tangent operators. All those issues and their solutions are identical to those already developed for rate-independent composites, e.g., elasto-plasticity.

Finally, and although as already mentioned, it is not rigorous to use the incremental formulation for ratedependent materials, experience has shown that there are cases where it leads to better predictions than the incrementally affine formulation. This issue is still not understood at the present time and investigations continue. In the mean-time, the incremental formulation is enabled in Digimat-MF for rate-dependent inelastic composite materials, whose phases obey EVP or VE-VP constitutive models.

Second-order Homogenization

Both the incremental and the incrementally affine formulations of MFH which were presented in previous sections are first-order. For inelastic composites, e.g., elasto-plastic or elasto-viscoplastic, this means that the comparison tangent operator of each phase is computed with the volume average of the strain field in the phase.

This mean value is called the first statistical moment of the per-phase strain field. In second-order homogenization, we use not only the first but also the second statistical moment of each phase's strain field. The second moment is linked to the variance. The latter enriching the statistical information compared to only a simple mean value, it is expected better predictions will be obtained using second-order instead of first order MFH.

In order to clarify and motivate the concept of second-order MFH, we consider a composite made of elastic–not necessarily linear– phases (r), each described by a strain energy function

$$W_r(\varepsilon(x)), \qquad \forall x \in \omega_r \tag{2-60}$$

A second-order Taylor expansion around a reference strain value, which is uniform in each phase, gives,

$$W_r(\varepsilon(x)), \approx W_r(\tilde{\varepsilon}_r) + \frac{\partial W_r}{\partial \varepsilon}(\tilde{\varepsilon}_r):(\varepsilon(x) - \tilde{\varepsilon}_r) + \frac{1}{2}\frac{\partial^2 W_r}{\partial \varepsilon \partial \varepsilon}(\tilde{\varepsilon}_r)::[(\varepsilon(x) - \tilde{\varepsilon}_r) \otimes (\varepsilon(x) - \tilde{\varepsilon}_r)], \tag{2-61}$$

$$\forall x \in \omega_r$$

We now define reference stress and stiffness values which are uniform in each phase.

$$\tilde{\sigma}_r \equiv \frac{\partial W_r}{\partial \varepsilon}(\tilde{\varepsilon}_r), \qquad \tilde{C}_r \equiv \frac{\partial^2 W_r}{\partial \varepsilon \partial \varepsilon}(\tilde{\varepsilon}_r) \tag{2-62}$$

Taking the per-phase volume average of the strain energy function, we find

$$\langle W_r(\varepsilon(x)) \rangle_{\omega_r} \approx W_r(\tilde{\varepsilon}_r) + \tilde{\sigma}_r : (\langle \varepsilon(x) \rangle_{\omega_r} - \tilde{\varepsilon}_r) + \frac{1}{2} \tilde{C}_r \langle (\varepsilon(x) - \tilde{\varepsilon}_r) \otimes (\varepsilon(x) - \tilde{\varepsilon}_r) \rangle_{\omega_r} \tag{2-63}$$

The average over all phases –weighted by the volume fractions– gives the effective (or macro) strain energy of the composite RVE

$$\overline{W}(E) = \sum_r v_r \langle W_r(\varepsilon(x)) \rangle_{\omega_r} \tag{2-64}$$

As for the reference strain, the simplest choice is to take the mean value in each phase. It then appears that the composite response depends on the *variance* of the per-phase strain field, which is related to the second statistical moment as follows:

$$\langle (\varepsilon(x) - \langle \varepsilon(x) \rangle_{\omega_r}) \otimes (\varepsilon(x) - \langle \varepsilon(x) \rangle_{\omega_r}) \rangle_{\omega_r} = \langle \varepsilon(x) \otimes \varepsilon(x) \rangle_{\omega_r} - \langle \varepsilon(x) \rangle_{\omega_r} \otimes \langle \varepsilon(x) \rangle_{\omega_r} \tag{2-65}$$

The above concepts have been extended to the incremental formulation of elasto-plastic composites whose phase's behavior is described by the J_2-plasticity model (with istropic hardening).

The current implementation in Digimat-MF shows that with respect to a first order formulation, the second-order theory brings significant improvement when following three conditions are met:

- fiber reinforcement
- high stiffness contrast between fibers and matrix
- the elasto-plastic matrix exhibits little hardening.

Otherwise, no significant differences are observed between the predictions of first- and second-order homogenization.

Since Digimat 6.0.1, a revised formulation has been developed in order to extend the 2nd order method to non J_2-plasticity models. For J_2-plasticity, both the original and the revised formulations are available. All three conditions are met for short glass fiber reinforced polyamide.

A typical example is illustrated in Figure 2-17.

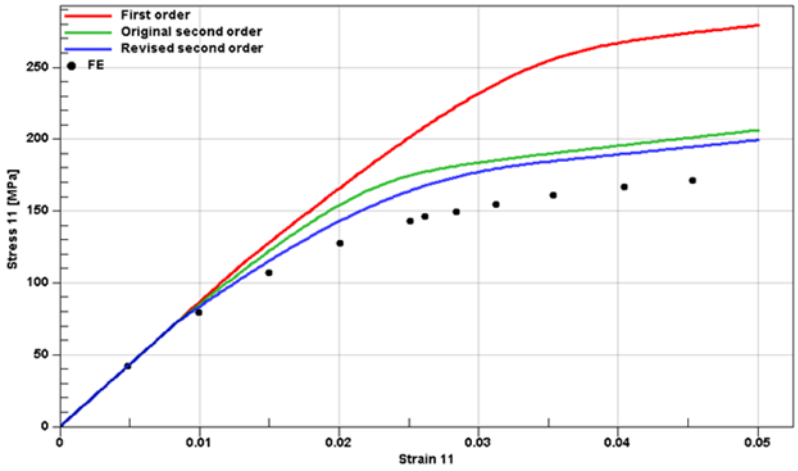


Figure 2-17 Short glass fiber reinforced polyamide. Fully aligned fibers. Uniaxial tension along the fibers' axis. Macro stress-strain results with: first-order MFH, second-order MFH original and revised formulation and direct FE on a RVE with 166 aligned fibers.

With respect to reference full-field FE results obtained on a RVE, it is seen from Figure 2-17 that first-order MFH greatly overestimates the macro response. The second-order MF formulation leads to significant improvement, but the results are still well off the reference FE targets. This is why a pragmatic approach is available in Digimat-MF: first-order incremental formulation with modified spectral isotropization.

The method is empirical and uses parameters which are obtained by fitting against stress/strain results at the composite level, obtained either with FE analysis or experimentally. Therefore, the modified spectral method is not truly predictive. However, it provides a practical workaround while research work continues in order to develop a fully predictive second-order formulation for elasto-plastic and elasto-viscoplastic composites.

Interaction Law Scheme

The homogenization model proposed by Mercier and Molinari (2009) is based on the exact solution of Eshelby's problem when an inclusion is embedded into an infinite reference medium. The inclusion and the reference material have an elasto-viscoplastic behavior. For this particular problem, Molinari (2002) proposed the following interaction law:

$$\dot{\varepsilon}_I - \dot{\varepsilon}^* = (a_0^{tg} - (P_0^{tg})^{-1})^{-1} : (\sigma_I - \sigma^*) + (a_0^{e} - (P_0^{e})^{-1})^{-1} : (\dot{\sigma}_I - \dot{\sigma}^*) \quad (2\text{-}66)$$

where $\dot{\varepsilon}^*$ denotes the remote loading prescribed on the reference medium at infinity and $\dot{\varepsilon}_I$ the strain rate in the inclusion phase. a_0^{tg} is the tangent viscoplastic stiffness tensor and a_0^{e} the

elastic stiffness tensor of the reference medium. P_0^{tg} (respectively P_0^e) denotes the Hill's tensor related to a_0^{tg} (respectively a_0^e). σ^* represents the Cauchy stress tensor at the remote boundary of the reference medium and σ_I the Cauchy stress tensor in the inclusion phase.

Let us consider a heterogeneous medium made of different phases having a nonlinear elasto-viscoplastic behavior and distributed in a disordered manner. The total strain rate $\dot{\varepsilon}$ of each phase is split into an elastic part $\dot{\varepsilon}^e$ and a viscoplastic part $\dot{\varepsilon}^{vp}$:

$$\dot{\varepsilon} = \dot{\varepsilon}^e + \dot{\varepsilon}^{vp} \tag{2-67}$$

The elastic part of the total strain rate is related to Cauchy stress tensor through an incremental elastic law:

$$\dot{\varepsilon}^e = (a^e)^{-1} : \dot{\sigma} \tag{2-68}$$

The viscoplastic part of the total strain rate is assumed to be volume preserving and is related to the deviatoric part of the Cauchy stress tensor by:

$$\dot{\varepsilon}^{vp} = \frac{\partial f}{\partial s} \text{ or } s = \frac{\partial g}{\partial \dot{\varepsilon}}(\dot{\varepsilon}^{vp}) \tag{2-69}$$

The tangent viscoplastic stiffness tensor is defined as follow:

$$a^{tg} = \frac{\partial^2 g}{\partial \dot{\varepsilon} \partial \dot{\varepsilon}}(\dot{\varepsilon}^{vp}) \tag{2-70}$$

For a Mori-Tanaka approach, the strain rate applied at the boundary of the reference medium is the one of the matrix phase. With this definition, it follows the relation between the strain rate in the inclusion phase and the strain rate in the matrix phase:

$$\dot{\varepsilon}_I - \dot{\varepsilon}_M = (a_M^{tg} - (P_M^{tg})^{-1})^{-1} : (\sigma_I - \dot{\sigma}_M) + (a_M^e - (P_M^e)^{-1})^{-1} : (\sigma_I - \dot{\sigma}_M) \tag{2-71}$$

The above formulation has been extended to thermo-viscoplastic analyses assuming the following strain decomposition:

$$\dot{\varepsilon} = \dot{\varepsilon}^e + \dot{\varepsilon}^{th} + \dot{\varepsilon}^{vp} \tag{2-72}$$

where the thermal strain ε^{th} is defined as follows:

$$\varepsilon^{th} = \alpha(T)[T - T_{ref}] - \alpha(T_{ini})[T_{ini} - T_{ref}] \tag{2-73}$$

The thermo-elastic part of the total strain rate is related to the time derivative of the Cauchy stress tensor through an incremental thermo-elastic law:

$$\dot{\sigma} = a^{e}:\dot{\varepsilon}^{the} + \dot{a}^{e}:\varepsilon^{the} + \dot{\beta} \quad \text{with} \quad \begin{cases} \dot{\varepsilon}^{the} = \dot{\varepsilon}^{e} + \dot{\varepsilon}^{th} \\ \beta = -a^{e}:\varepsilon^{th} \end{cases} \tag{2-74}$$

The implementation of this formulation in Digimat-MF shows that, with respect to a discrete affine formulation, the interaction law scheme brings significant improvements when one of the following three conditions is met:

- creep predictions
- cyclic and monotonic loadings
- non-elastic inclusions

Homogenization under Finite Transformation

In this section, we consider the homogenization of composites (such as rubber matrix composites) under finite or large transformation, i.e., finite strains, rotations and displacements. We first need to introduce some notation and recall some results for homogeneous solids.

Consider a solid body (which in this section designates a RVE or a part of it) which occupies a domain Ω_0 (with boundary $\partial\Omega_0$) in the reference configuration and a domain ω_t (with boundary $\partial\omega_t$) in the current configuration at time $t > 0$. A fixed Cartesian frame is considered. A material particle is determined by its position vectors X and x with respect to Ω_0 and ω_t so that for a motion $\varphi(X, t)$, we have

$$x = \psi(X,t); \text{ or equivalently } x_i = \psi_i(X_A,t);\ A, i - 1,2,3. \tag{2-75}$$

The convention of upper case indices for X_A and lower case x_i will be used throughout. The deformation gradient and its determinant are defined as follows:

$$F(X,t) = \frac{\partial x}{\partial X} = GRAD\ x\,,\ J(X,t) = \det\ F(X,t) \tag{2-76}$$

Different stress measures are needed: the first Piola-Kirchhoff (P-K) stress P (with components P_iA), whose transpose is the nominal stress $P^n = P^T$, the second P-K stress S (with components S_{AB}), the Cauchy (true) stress σ (with components σ_{ij}), the Kirchhoff stress τ (with components τ_{ij}). Stresses S, σ and τ are symmetric. The following relations between the different measures hold

$$P = F \cdot S, \qquad \tau = J\sigma = P \cdot F^T = F \cdot P^T = F \cdot S \cdot F^T \tag{2-77}$$

We consider that each phase's material obeys a hyperelastic constitutive model defined by a strain energy function with respect to Ω_0 which satisfies a material objectivity condition

$$W(X,F) = W(X, Q \cdot F) = \hat{W}(X,C) \tag{2-78}$$

where Q is an arbitrary rigid rotation superposed on ω_t and $C = F^T \cdot F$ the Right Cauchy-Green strain.

Stresses P^n and S are obtained from the stress/strain relations

$$P^n = \frac{\partial W}{\partial F}, \; S = 2\frac{\partial \hat{W}}{\partial C} \tag{2-79}$$

Stresses τ and σ can then be computed from the above equations.

Different tangent operators can be defined:

- a material tangent operator Γ (with components Γ_{ABCD}) such that

$$\dot{S} = \Gamma : \frac{\dot{C}}{2}; \; \Gamma = 4\frac{\partial^2 \hat{W}}{\partial C \partial C}; \; \Gamma_{ABCD} = \Gamma_{BACD} = \Gamma_{ABDC} = \Gamma_{CDAB} \tag{2-80}$$

- a spatial elasticity tensor Υ (with components Γ_{ijkl}) such that

$$\dot{\tau} - 1.\tau - \tau.1^T = \gamma : d; \; \gamma_{ijkl} = \gamma_{jikl} = \gamma_{ijlk} = \gamma_{klij} \tag{2-81}$$

- a nominal tangent operator Λ (with components Λ_{AiBj}) such that

$$\dot{P}^n = \Lambda : \dot{F}; \; \Lambda = \frac{\partial^2 W}{\partial F \partial F}; \; \Lambda_{AiBj} = \Lambda_{BjAi} \tag{2-82}$$

In the above notation, a superposed dot designates a material time derivative, l (with components l_{ij}) is the velocity gradient and d (with components d_{ij}) its symmetric part.

Important remarks are that Γ is defined with respect to Ω_0, Υ with respect to ω_t, and both have minor and major (diagonal) symmetries, while Λ has indices with respect to both Ω_0 and ω_t, and possesses diagonal symmetry only.

We now consider the case of composite materials, a first-order incremental formulation proposed by Nemat-Nasser and Hori (1993). The formulation is based on the following three basic ideas. First perform volume averaging on reference configuration. Second, use deformation gradient F, nominal stress P^n and their rates. Third, work with rate constitutive equations under the nominal format, i.e., with the nominal tangent operator.

Consider the case of linear boundary displacements and velocities

$$x = F^0 \cdot X; \; \dot{x} = \dot{F}^0 \cdot X \text{ on } \partial\Omega_0 \tag{2-83}$$

with F^0 and its time derivative uniform on $\partial\Omega_0$. The following volume averaging results are found

$$\langle F \rangle_{\Omega_0} = F^0; \; \langle \dot{F} \rangle_{\Omega_0} = \dot{F}^0 \tag{2-84}$$

Consider now the case of uniform boundary traction and traction rates

$$\mathbf{T} = \mathbf{P}^0 \cdot \mathbf{N}\,;\ \dot{\mathbf{T}} = \dot{\mathbf{P}}^0 \cdot \mathbf{N} \text{ on } \partial\Omega_0 \tag{2-85}$$

with $\mathbf{P}^0$ and its time derivative uniform on $\partial\Omega_0$. The stress averaging results hold

$$\langle \mathbf{P}^n \rangle_{\Omega_0} = (\mathbf{P}^0)^T; \qquad \langle \dot{\mathbf{P}}^n \rangle_{\Omega_0} = (\dot{\mathbf{P}}^0) \tag{2-86}$$

The averaging results show that the main homogenization problem in the incremental formulation is to relate the average of the nominal stress rate and the average of the time derivative of the deformation gradient over the reference configuration, i.e., the volume average over the RVE of the micro fields. This can be accomplished by finding a suitable expression of a macro nominal tangent operator Λ_{mac} such that

$$\langle \dot{\mathbf{P}}^n(\mathbf{X}, t) \rangle_{\Omega_0} = \Lambda_{mac}(t) : \langle \dot{\mathbf{F}}(\mathbf{X},t) \rangle_{\Omega_0}, \text{ i.e. } \langle \dot{P}^n_{Ai} \rangle_{\Omega_0} = (\Lambda_{mac})_{AiBj} \langle \dot{F}_{jB} \rangle_{\Omega_0} \tag{2-87}$$

In order to do so, we associate with each phase a fictitious comparison medium whose tangent operator is uniform. With this notion, Nemat-Nasser and Hori (1993) showed that it is possible to generalize the results of Eshelby as well as mean-field homogenization models to finite strain rate-independent inelasticity.

In Digimat-MF, the above formulation was enriched, developed and implemented for the homogenization of composite materials under complete finite transformation in two cases: (a) hyperelastic materials in any phase, (b) Leonov-EGP elasto-viscoplastic material model in the matrix phase. For other inelastic composites, e.g., J_2-plasticity or J_2-viscoplasticity, it is only possible in the current version to simulate large rotations but small strain, when coupling Digimat-MF with a finite element solver.

Taylor Homogenization

A simple homogenization scheme for polycrystalline aggregate is proposed in Digimat-MF. This scheme is based on the classical Taylor (iso-strain-rate) averaging scheme and applies only to crystal plasticity materials.

3 Graphical User Interface

- General Overview
- Batch Mode
- Setting Dependencies

General Overview

This section of the documentation focuses on presenting the graphical user interface (GUI) of Digimat-MF, the Digimat product giving the opportunity to test composite material at the representative volume element level (RVE). To do so, not only the material properties definition is required but also the RVE microstructure as well as the loading to apply on the RVE boundaries.

The full process, from the definition of the composite material until the post processing of the results, is presented in this section of the documentation.

Icons Definition

Figure 3-1 lists all the available icons in the top toolbar of the GUI.

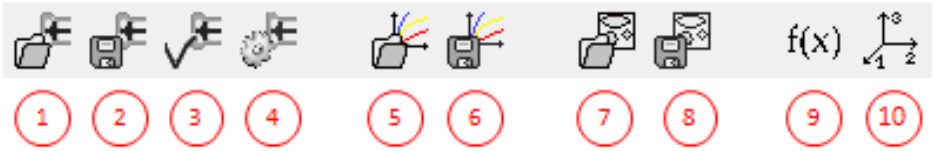


Figure 3-1 List of all icons available inside Digimat-MF.

1. Load analysis file (`.daf` or `.mat`)
2. Save analysis file (`.daf`)
3. Perform a data check
4. Run the current analysis
5. Load a material definition
6. Save a material definition
7. Load a phase definition
8. Save a phase definition
9. Define a function
10. Define a new axis system

Analysis Definition

The definition of a Digimat-MF analysis requires the completion of several steps. These steps are highlighted in the Digimat tree, whose items should be defined from top to bottom for the analysis to be completely defined.

Actions in the tree can be performed in three different ways:

- Right-clicking on a tree item makes a context menu appear, from which the action to be performed can be chosen.

- Clicking on the appropriate item in the top menu bar and choosing the action to carry out.
- Clicking on one of the icons in the icon bar.

To ease the validation of an analysis, the symbols in front of the different tree items turn bright when the corresponding item is correctly created and hence valid. Each dark symbol represents an invalid or not yet validated item and will lead to an error message during data check or at job submission.

Pre-processing

The interface window of Digimat-MF is split into three areas: the menu bar combined with the toolbar at the top, the Digimat tree on the left and the main area on the right.

To define an analysis in a complete and successful way, all analysis items of the Digimat tree need to be defined, from top to bottom. Each of them is described in the following paragraphs.

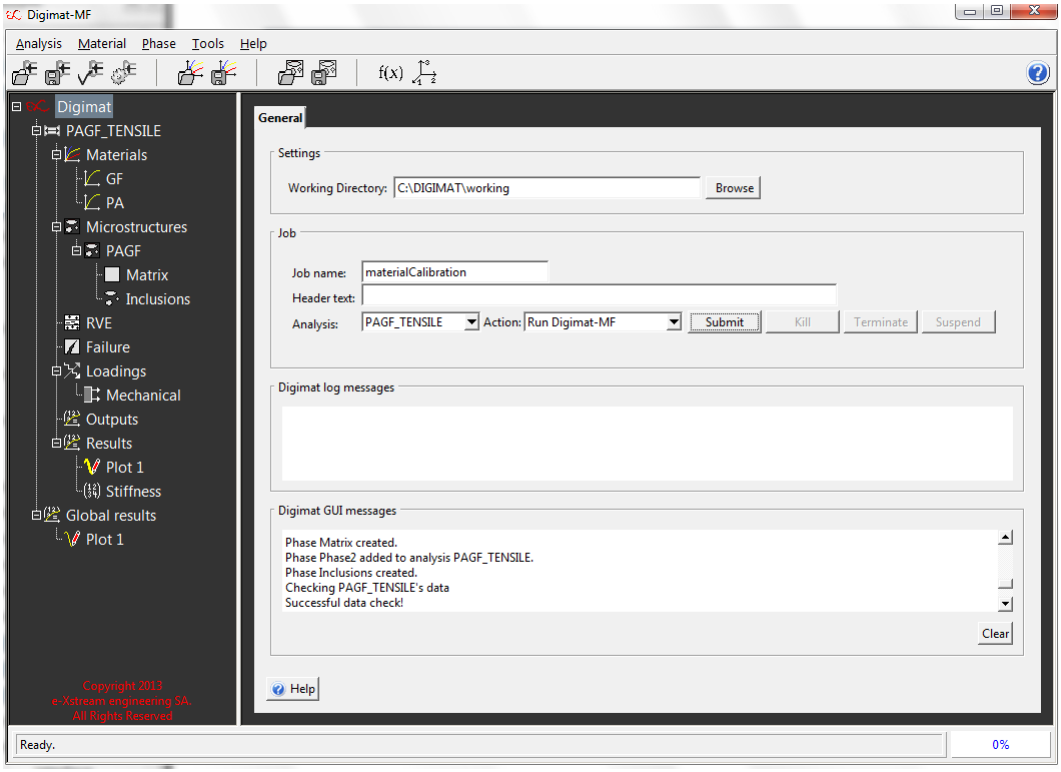


Figure 3-2 Graphical user interface of Digimat-MF.

Digimat Item

Digimat is the first item in the Digimat tree. Clicking on Digimat opens a tab called **General** where:

- The **working directory** of the analysis can be specified. This directory is where all results files are stored. By default, this directory is the working directory of Digimat.
- The **Job Name** of the analysis can be given. This name, together with the analysis name, is used to produce the full name of the generated results files, which are designated by a concatenation of the job and the analysis names.
- A **header** section allowing the user to add information about the job can be defined. It is a free writing field that has no effect on the computation.
- The current **analysis** can be selected, which corresponds to the one on which actions will be performed when requested.
- An action to be performed can be selected:
  - **Run Digimat-MF** starts the computation.
  - **Data check** verifies whether the analysis is correctly defined or not. This action can also be performed by clicking on the icon.
  - **Write file** saves in the working directory a file containing all the analysis definition. The extension of this file is .mat.
  - **View file** opens the working directory and allows the user to select the file to be edited.

As soon as an action is chosen, the **Submit** button must be hit to execute it. Submitting a job can also be done by clicking on the appropriate icon (see Figure 3-2). Running jobs can also be killed, suspended or terminated.

As mentioned earlier, the name of the results files is the concatenation of the **Job Name** and the **Analysis name**. By default, it is DefaultJobName_Analysis1.

A specific area is dedicated to the output of informative messages, called the **Digimat log messages**. Those are written and updated by Digimat throughout all the computation process.

A second area is dedicated to the output of the information messages sent by the GUI. This area summarizes all the actions performed by the user in the GUI.

Analysis1

Clicking on **Analysis1** opens a new tab called **General parameters**. A second tab called **Integration parameters** is also available.

Analysis1 is a generic name indicating where the definition of an analysis starts in the tree. Digimat supports multiple analysis definitions. The Digimat tree can then contain more than one analysis. By default the name Analysis1 will be incremented to Analysis2, Analysis3, . . .

Adding a new analysis in the tree can be easily done by right-clicking on the **Digimat** item. The name Analysis1 can be changed by modifying the name field in the **General parameters** tab. Switching to Digimat-FE is also possible by changing the material modeler.

The General parameters tab is composed of three main areas allowing the user to define the general options of his analysis.

Mean-Field Homogenization (MFH) This option is to be switched off if the material computation is to be carried out on a homogeneous material RVE. In such a case, only the matrix phase will be considered during the computation.

When switched on, MFH is used to predict the behavior of the RVE subject to a certain loading. All options relative to the homogenization procedure are described in detail in the Mean-field Homogenization Theory section.

The second order homogenization is selected by default. If the analysis that you prepare does not support second order, Digimat will automatically switch to the first order homogenization.

Analysis type Five analysis types can be performed in Digimat-MF: mechanical, thermomechanical, thermal, electrical and coupled thermal mechanical. The chosen analysis type determines the material models available to the user during the analysis definition.

The goal of thermal and electrical analyses is to compute the thermal or electrical conductivity of the composite. The goal of the coupled thermal mechanical analysis is to compute both the mechanical properties and the thermal conductivity of the composite.

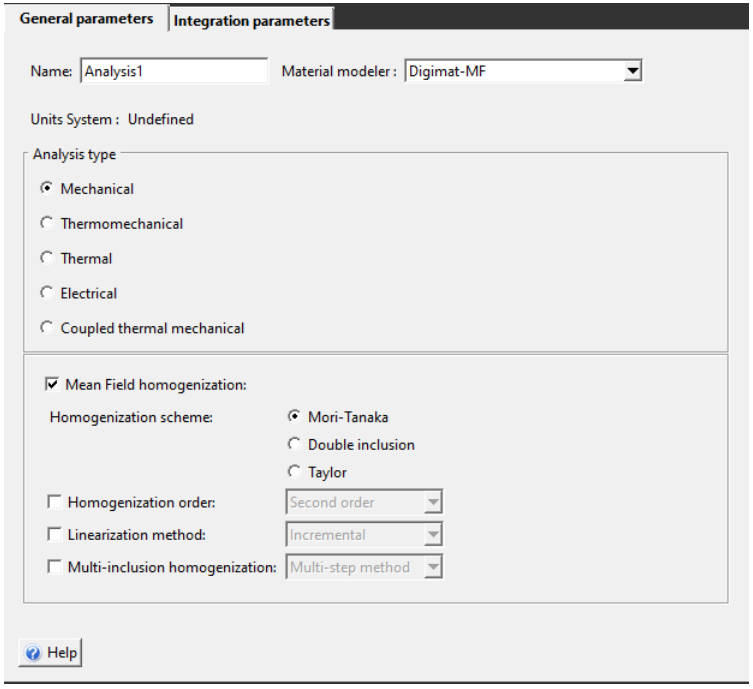


Figure 3-3 General parameters in Digimat-MF.

Geometrical nonlinearities This option must be toggled on when working with finite transformations. This is the case with hyperelastic material models and the Leonov-EGP one. All other models are valid in the framework of small strain.

Integration parameters In the integration parameters tab, control on the following parameters is given:

- The time stepping parameters of the analysis.
- The tolerances for the homogenization scheme and loading controls.
- The level of the integration scheme.
- The option for the orientation tensor discretization.

Material

The **Material** item is parent to all the materials present in the composite. Adding a material can be done by right-clicking on the Material item or by going to the menu **Material** → **New**. As soon as a new material is added, two tabs are then displayed in the main area of the GUI: the **Model** and the **Parameters** tabs.

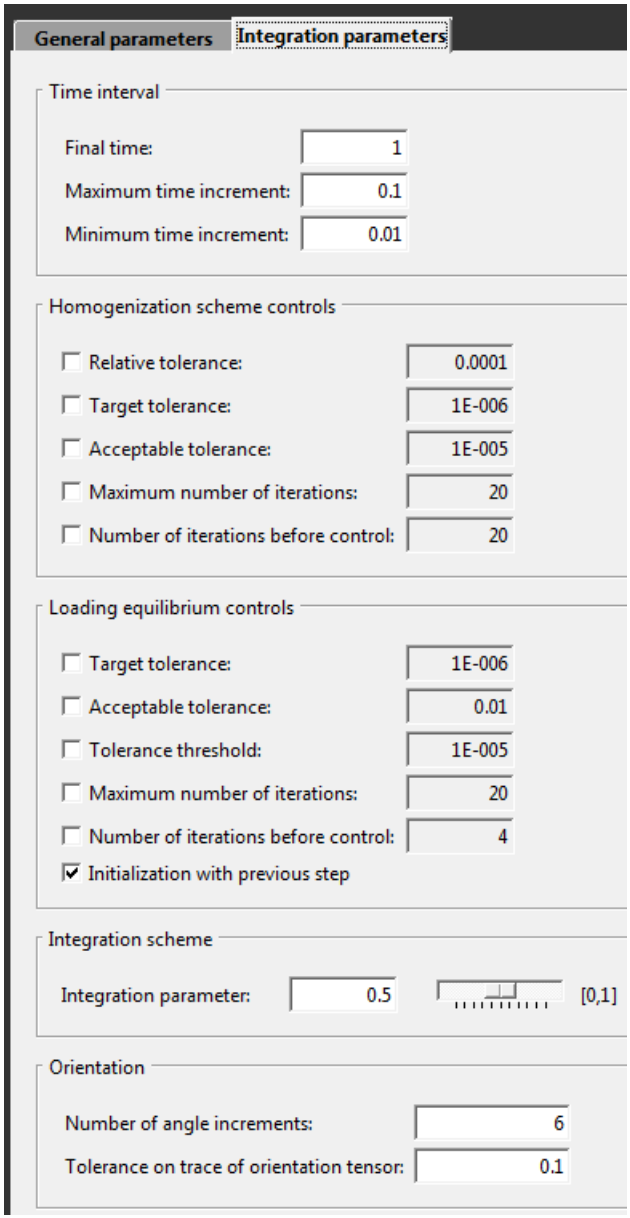


Figure 3-4 Integration parameters in Digimat-MF.

To move from the **Model** tab to the **Parameters** one, hit the validate button in the bottom right corner. The generic name of a material is Material1 where the number is incremented for each new material. Of course, this name can be changed.

In the **Model** tab, the constitutive law relative to the material to be modeled is to be selected. The **Parameters** tab then allows the definition of the relevant material parameters. A material can be loaded, copied, saved or deleted. These actions are available either from the material context menu (right-click on the Material tree item), the Material item in the top menu bar or the icon bar (see Figure 3-2).

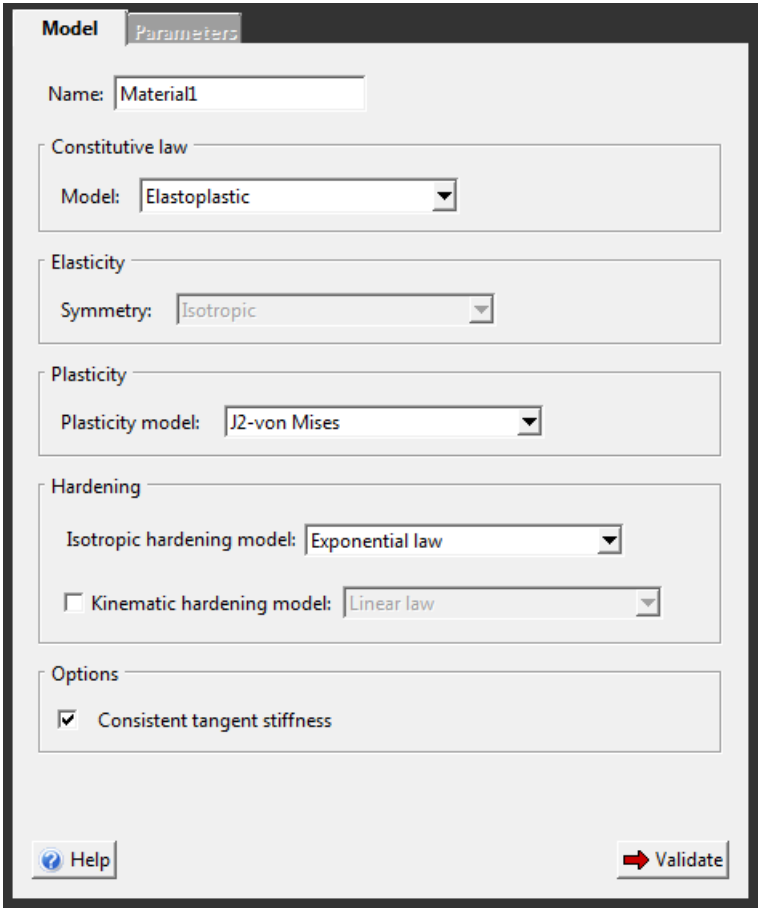


Figure 3-5 Model tab of the Material tree item.

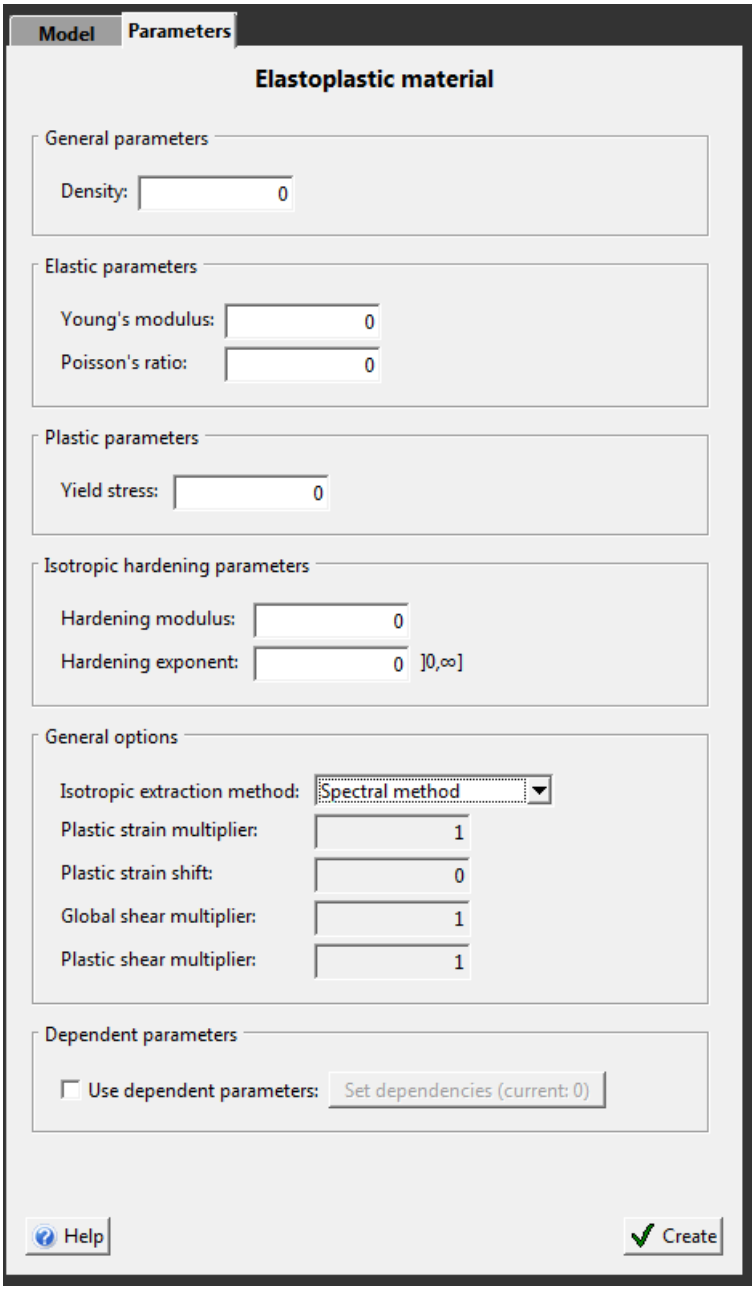


Figure 3-6 Parameters tab of the Material tree item.

Microstructure

The objective of the microstructure section is to answer at the following questions:

- What material is the matrix?
- What material is the filler?
- What is the amount of filler in the matrix?
- What is the shape of the inclusions?
- What is the orientation of the inclusions?
- Are there any voids present inside the matrix?

By right-clicking on the **Microstructure** tree item, a new microstructure can be added to the Digimat tree. The generic name for microstructure is **Microstructure1** where the number 1 is incremented for each new microstructure. By clicking on the created microstructure, it is possible to choose between a generic microstructure and some special ones: fabrics and lattices. Below each microstructure, several phases can be added. If defining a mono-layer composite, several microstructures can be defined inside the analysis but only one of them will be used in a single Digimat-MF analysis. If defining a multilayer composite (see General Overview), then the different microstructures can be used to describe each layer of the layered RVE. One microstructure can be assigned to each layer.

A microstructure can be copied or deleted. Phases can be loaded, saved or deleted. As for the materials, these actions are available from the context menu (right-click on the Microstructure item), the top menu bar or the dedicated icons.

Several types of phases exist for each type of microstructure:

- Generic microstructure: matrix, inclusion, continuous fiber, void
- Other microstructures (fabrics and lattices): matrix, yarn (fabrics only), inclusion and void

In one single microstructure, at least one matrix phase must be defined.

Matrix phase

The definition of the matrix phase solely consists in the assignment of a material to this phase.

Inclusion phases

For inclusions, at least two tabs must be filled: the Type and the Parameters tabs. In the Type tab, the inclusion phase must be assigned a numerical behavior as well as the constituent material. If a coating layer exists around the inclusions, it also has to be defined.

After validation of the phase type, various parameters relative to the inclusion phase must be defined: phase fraction, inclusion aspect ratio and their orientation.

If there is coating layer around the inclusions, a third tab called Coating becomes available. The coating material is to be assigned in this tab, as well as the coating fraction.

Void type

If the inclusion phase corresponds to cavities, the void phase type should be used. No material property needs to be associated with such a phase type.

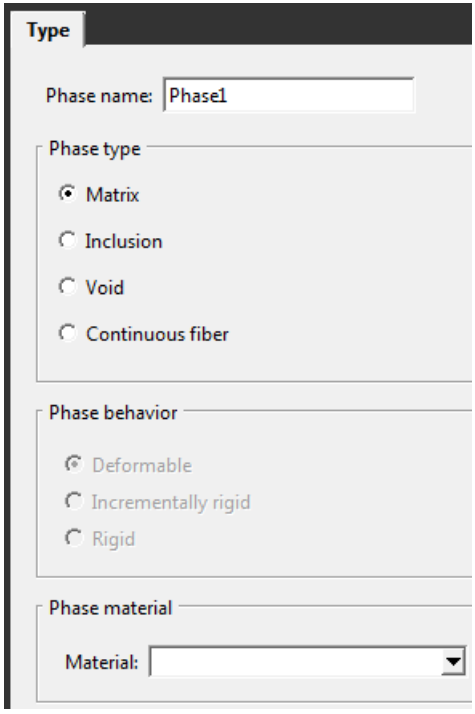


Figure 3-7 Definition of the type of inclusion.

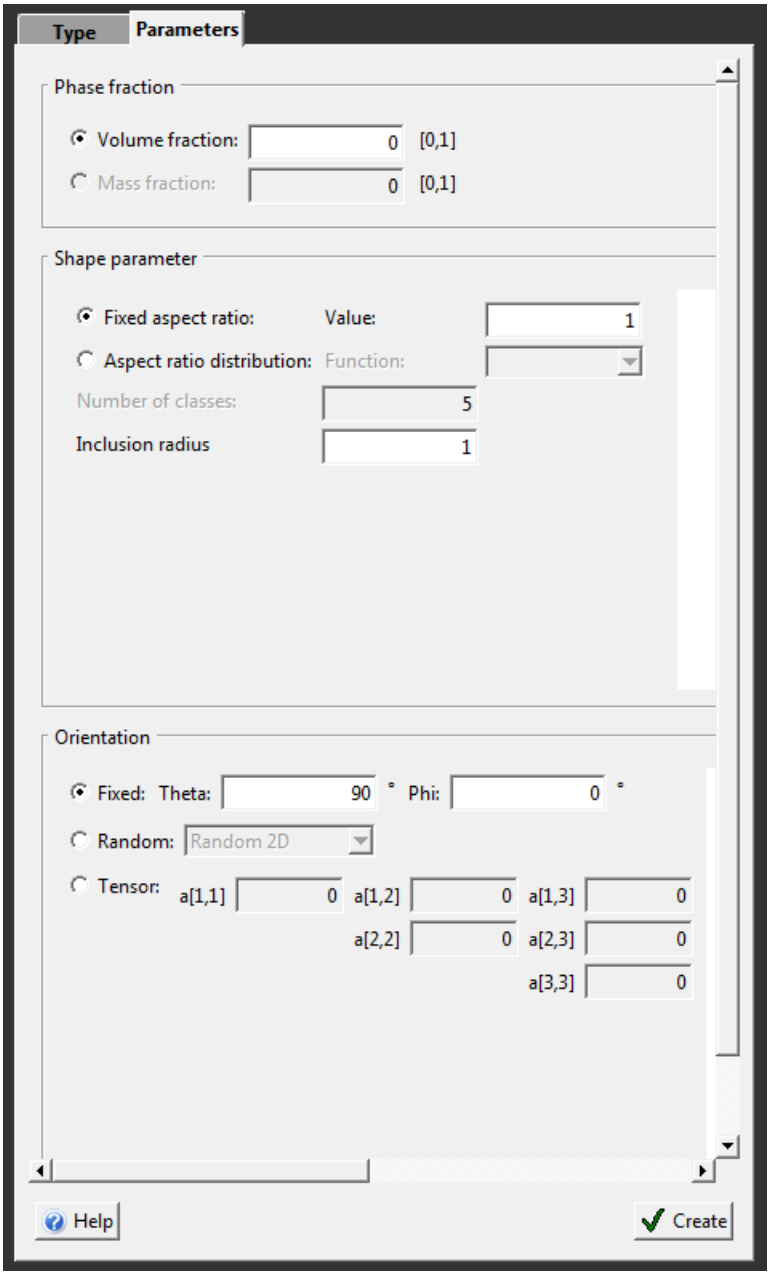


Figure 3-8 Definition of the phase fraction, aspect ratio and orientation of the inclusion phase.

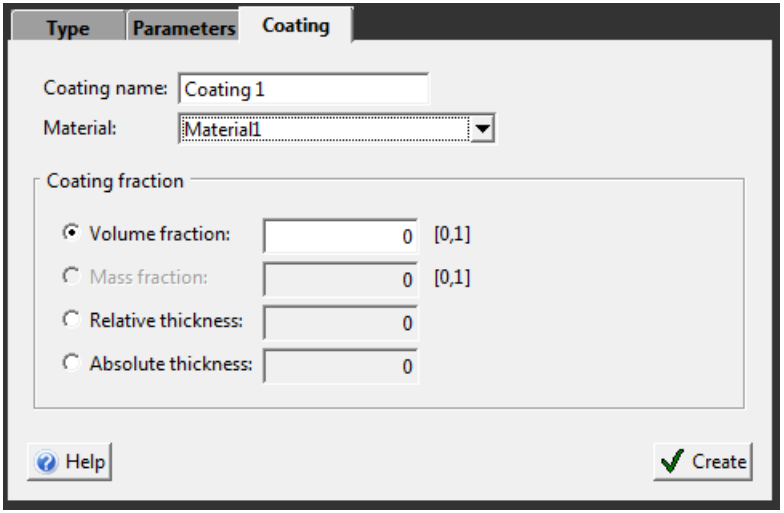


Figure 3-9 Coating definition.

RVE

The representative volume element (RVE) represents the material point on which boundary conditions are applied. Only one RVE can be defined in a Digimat-MF analysis. A RVE can be composed of one or several layers. When a multilayer structure is defined, each layer can have its own microstructure or the same microstructure can be used with different orientations for the inclusions. The thickness is another parameter that can vary from one layer to another.

A multilayer RVE can be defined manually or its characteristics can be imported from Moldflow/Midplane orientation files (see Method 3: Import from Orientation File). This allows describing the evolution of the inclusions' orientation through the thickness (from the top to the bottom) of the composite microstructure. This can significantly improve the prediction of the Digimat

response as the fiber orientation evolution throughout the thickness is modeled more accurately. Multilayer RVE should be used whenever possible.

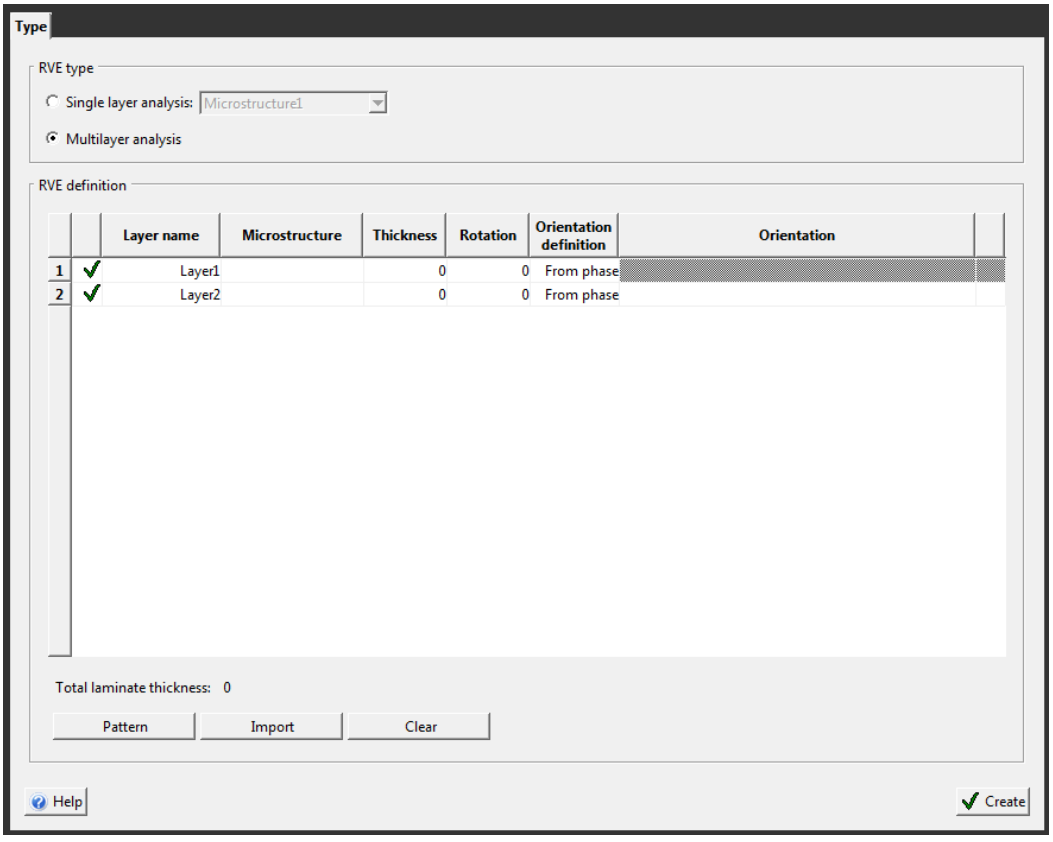


Figure 3-10 Multilayer RVE definition.

Failure

The failure section is used to define and assign failure indicators. Failure indicators are not mandatory in a Digimat-MF analysis.

Clicking on the **Failure** item displays the failure indicator assignment panel in the right area of the GUI (see Figure 3-11).

Adding a failure indicator can be done by right-clicking on the **Failure** item in the tree. Once a new failure indicator is added, a new tab is displayed in the left area of the GUI. The generic

name of a failure indicator is FailureIndicator1 where the number is incremented for each new failure indicator.

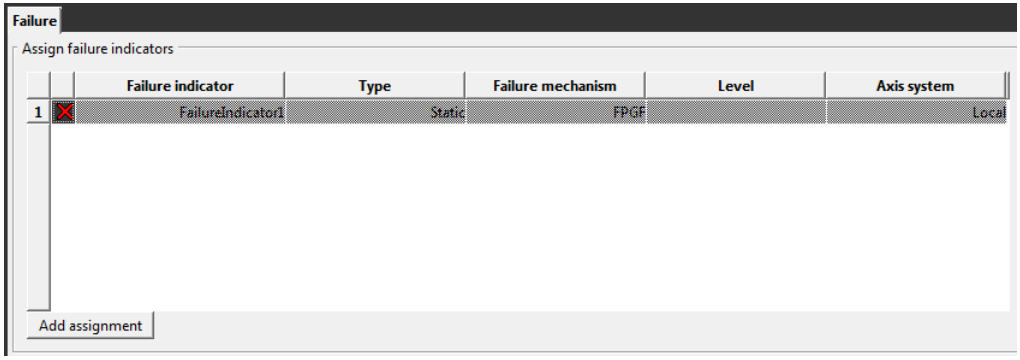


Figure 3-11 Failure indicator assignment.

This name can be changed. In this new tab (see Figure 3-12), the relevant model for the failure indicator can be chosen, and the different required parameters must be specified. If strain rate dependencies are defined, a second tab appears, allowing their definition (see Figure 3-13).

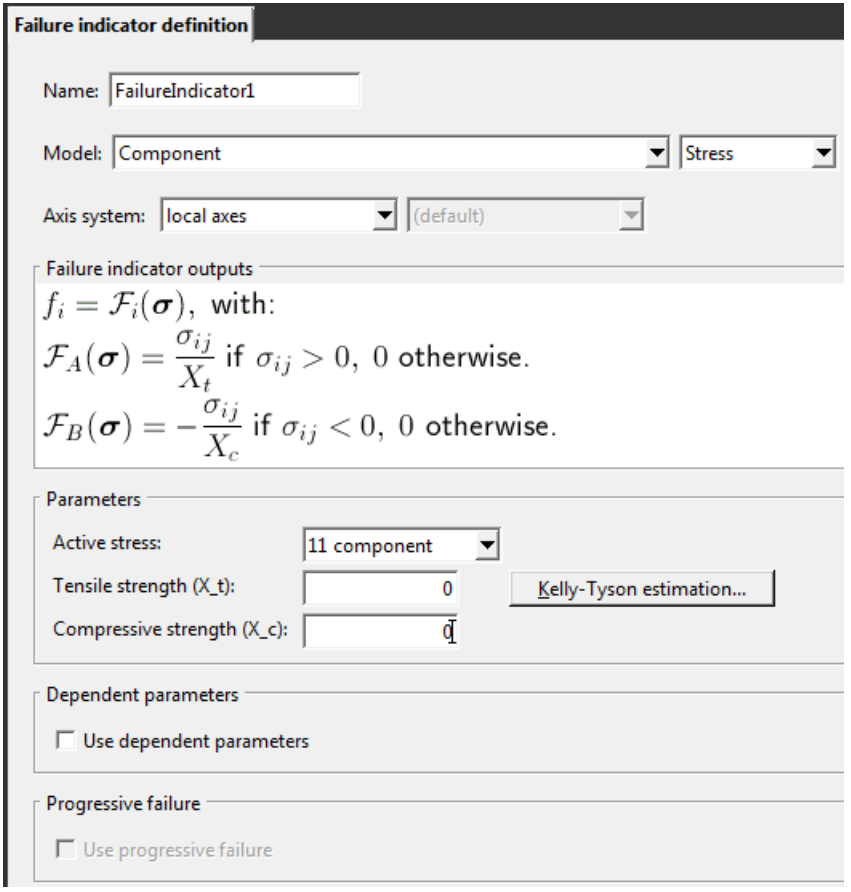


Figure 3-12 Failure indicator definition.

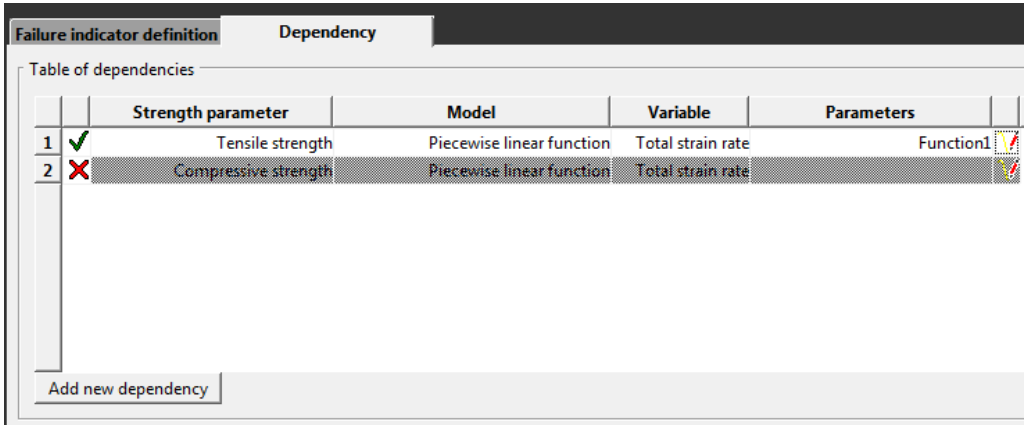


Figure 3-13 Strain rate dependency definition.

Loadings

A Digimat-MF analysis requires the definition of the loadings to be applied on the RVE boundaries. The definition is performed in two steps. In the first tab, the loading type is to be specified. The following options are available:

- Mechanical loading: the type of loading (stress or strain and the direction of loading) and its time history definition must be specified.
- Thermal and electrical loadings: only the time history option needs to be chosen.

In the second tab, the load amplitudes need to be specified. The parameters to be input depend on the selected loading type.

In the case of thermo-mechanical analyses, both a mechanical and a thermal loadings need to be defined.

Computation

To start a computation in Digimat-MF, one must:

- Select the analysis to be solved. This can be done by clicking on the title of the analysis in the Digimat tree or by selecting the correct analysis from the available list, located in the General parameters tab (see section about Digimat item above).
- Select the **run Digimat-MF** action in **General parameters** tab and click **submit** button, or on the corresponding icon.

The log file is echoed in the **General parameters** tab and can be used to follow the progression of the computation.

Post-processing

The post-processing of Digimat-MF results is done under the **Results** item in the Digimat tree. Three types of actions are available: plotting results data, getting the elastic stiffness or the conductivity values.

Plot

By clicking on a plot item, the plot area appears in the main window. Plotting results requires one to load them first in the plot item. This can be done by right-clicking on a plot item and selecting **Load results** from the context menu. The loaded files depend on the selected analysis. Results at the composite land per-phase levels are loaded when dealing with heterogeneous RVE. Only the homogeneous response is loaded in case of a homogeneous RVE. Loaded results appear as a child item to the parent plot item. The date and the time at which the analysis was run are also indicated. This allows distinguishing different results obtained from the same analysis.

Additional actions on a plot item are available from the context menu:

- **Load functions**: to load functions in the plot area and be able to plot them in the viewport.

- **Clear plot**: to erase the current plot and results from the plot area.
- **Clear results**: to erase the loaded results of the selected plot item without clearing the plot area.

The control of the plot area is done via several buttons:

- **Create plot** button allows you to add curves in plot area. By clicking on Create plot, a new window pops up, listing all the results that can be plotted.
- **Edit plot** allows one to replace or to remove curves.
- **Import data** allows the importation of curves from a text file. This can be used to import experimental data to compare them with Digimat-MF prediction.
- **Load results** file is used to load results obtained in a previous analysis without have to load the complete analysis.
- **Edit properties** gives access to a menu to customize your curve, e.g., to change color, to add symbols on curve or to change the name in the legend.
- **Range** is used to adapt the ranges of X and Y-axes.
- **Legend** gives access to options controlling the position and the size of the legend.
- **General option** allows one to control the thickness of the plot, the size of the symbol and the type of X-axis (linear or log).
- **Axis title** is used to customize the title of the plot area, the name of X and Y-axes and the number of label along X and Y-axes.
- **Export plot** saves the content as a picture (bmp, jpeg, jpg, tiff, png, eps, ps) or an ascii file (txt).

Some of these actions are also available by right-clicking in the plot area. The Show/Hide button can be used to hide the controls from the plot area.

Several plot areas can be used in one Digimat-MF session. Additional plot items can be created via the context menu of the **Results** item.

Stiffness / Conductivity

Stiffness and Conductivity areas are very similar.

As for the plot area, results must first be loaded before they can be accessed. This can be done via the context menu of the Stiffness (Conductivity) tree item. Again, the loaded results are the ones related to the currently selected analysis. Note that these results are stored in the `.eng` file in the working directory.

Batch Mode

Both the GUI and the Digimat-MF solver can be invoked using a command prompt, or can be driven using a script. Here is the syntax to be used (use absolute paths if necessary):

- To load a Digimat analysis file in the graphical user interface, type:

  ```
  digimatGUI.exe input=Analysis.daf
  ```

- To run a Digimat-MF computation, type:

  ```
  digimat.exe input=Analysis.mat
  ```

Remark: When executing Digimat in your command-line session, the program might fail if your PATH system variable is incorrectly set. If this happens, the required directories are given in the batch file (*.bat) located near the executable; you can add them to your PATH system variable, or create a featured batch file (recommended in the case of multiple installations).

Setting Dependencies

Dependencies can be set in Digimat for the following types of parameters:

- mechanical and failure properties depending on the temperature
- mechanical and failure properties depending on the strain rate
- user-defined loading histories
- mechanical and failure properties depending on user-defined variables.

The definition of dependencies can be done via the definition of a piecewise linear function (also refer Functions on how to work with functions within the Digimat platform) or by selecting one of the available predefined dependency types.

Temperature-dependent Mechanical Properties

When performing thermo-mechanical analyses in Digimat, the mechanical properties can be defined as temperature- dependent, i.e., they vary with the temperature. In fact, it is advised to use temperature-dependent properties when broad ranges of temperature are involved in an analysis.

The definition of temperature-dependent mechanical parameters requires the previous definition of a function (see Functions). This function being defined, it is available for scaling the initial mechanical parameter, as described hereafter:

$$\Lambda(T) = \Lambda_0 f(T) \tag{3-1}$$

with Λ the scaled mechanical parameter, 0 denoting the initial value and T the temperature.

To set a dependency **over a material modulus** (e.g. in the Material Parameters tab), the user should first select the parameter to be scaled, the independent variable and the function to use, and then hit **Add** button.

Dependencies can be enabled, disabled and deleted by clicking on the corresponding buttons.

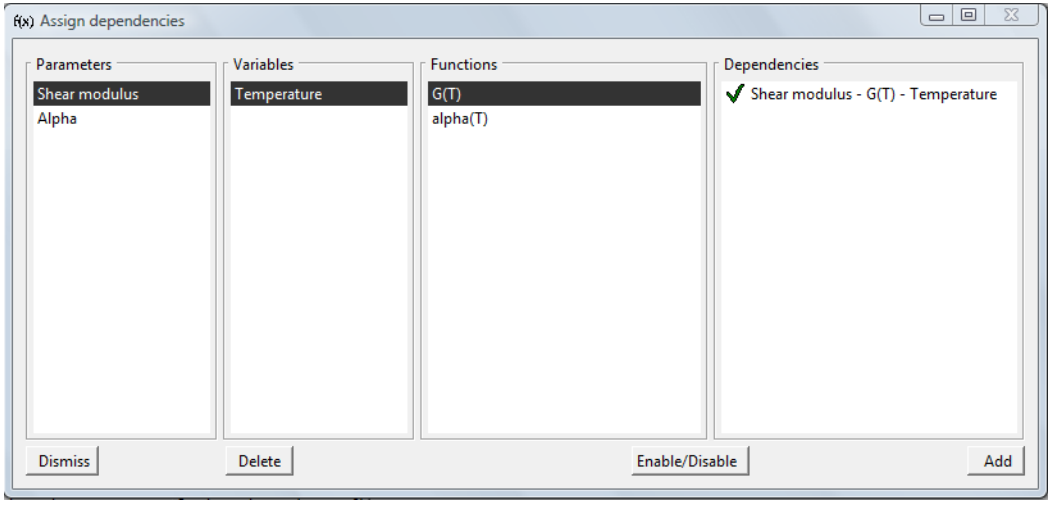


Figure 3-14 Setting temperature dependencies over mechanical moduli.

To set a dependency **over a failure parameter moduli** (e.g., in the **Failure Indicator tab**), the user should first check the box **Use dependent parameters** in the **Failure indicator definition** tab, then click on the **Create** button. This opens the **Dependency** tab, in which a table enables one to define one or several dependencies over each failure parameter. Please refer Setting Dependencies Over Strength Parameters for more information on this subject.

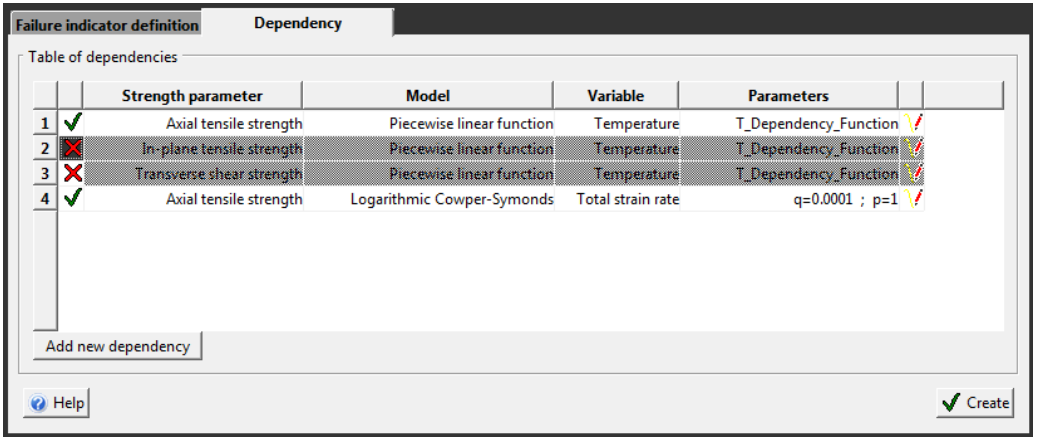


Figure 3-15 Setting temperature dependencies over failure parameters.

Note that functions are not extrapolated outside their domain of definition, they are assumed constant. Digimat will use the closest value in the domain of definition.

Strain Rate-dependent Mechanical Parameters

Two types of mechanical parameters can depend on the strain rate:

1. The yield stress of a viscoelasto- viscoplastic material (see Viscoelasticity-viscoplasticity)
2. The strengths of a failure criterion (see Failure Indicators).

The procedure to set the rate dependency is identical in both cases.

Three types of scaling are available to model the rate dependency of mechanical parameters:

- the Cowper-Symonds scaling law
- the logarithmic Cowper-Symonds scaling law
- an user-defined piecewise linear function

The two Cowper-Symonds laws are defined as follows:

$$\Lambda(\dot{\varepsilon}) = \Lambda_0\left[1+\left(\frac{\dot{\varepsilon}}{\dot{\varepsilon}_{ref}}\right)^{1/p}\right], \Lambda(\dot{\varepsilon}) = \Lambda_0\left[1+\left(\log\frac{\dot{\varepsilon}}{\dot{\varepsilon}_{ref}}\right)^{1/p}\right] \quad (3\text{-}2)$$

while a piecewise linear function scales the mechanical parameter as follows:

$$\Lambda(\dot{\varepsilon}) = \Lambda_0 f(\dot{\varepsilon}) \quad (3\text{-}3)$$

The Cowper-Symonds parameters are the exponent and the reference strain rate, respectively, while the 0 denotes the initial value of the scaled mechanical parameter Λ. The scalar strain rates are computed as the norm of the rate of the (total or plastic) strain tensor, as follows:

$$\dot{\varepsilon} = \sqrt{\frac{2}{3}\dot{\underline{\varepsilon}}:\dot{\underline{\varepsilon}}} \quad (3\text{-}4)$$

Remark: This expression yields a value which is distinct from the $\dot{\varepsilon}_{11}$ value of an uniaxial loading, although both definitions do match when the tangent behavior is incompressible (e.g., when deformation is purely plastic and incompressible).

To set a strain rate dependency **over a material modulus** (e.g., in the Material Parameters tab), the user should first select the parameter to be scaled, the independent variable, the scaling model, specify the required parameters or the appropriate scaling function, and then hit the **Add**

button. Dependencies can be enabled, disabled and deleted by clicking on the corresponding buttons.

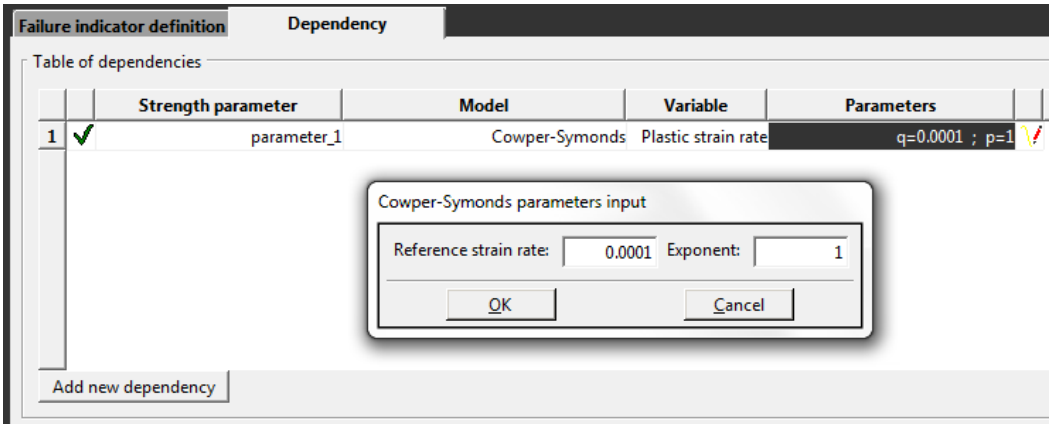


Figure 3-16 Setting strain rate dependencies.

To set a strain rate dependency **over a failure parameter moduli** (e.g. in the Failure Indicator tab), the user should first check the box **Use dependent parameters** in the **Failure indicator definition** tab, then click on the **Create** button. This opens the **Dependency** tab, in which a table enables one to define one or several dependencies over each failure parameter. The three scaling models ("Cowper-Symonds" and "Logarithmic Cowper-Symonds" and "Piecewise linear") are the same as previously discussed. Refer Setting Dependencies Over Strength Parameters for more information on this subject.

Note that functions are not extrapolated outside their domain of definition, they are assumed constant. Digimat will use the closest value in the domain of definition.

Time-dependent Loading Histories

Complex loadings can be specified by the user, using the user-defined loading history capability of Digimat (see Loadings section). The scaling can be applied to the loading peak values specified for any loading.

Only function scaling is available to define time-dependent loading histories. The procedure is similar to the setting of temperature-dependent mechanical properties. Refer to the corresponding section for more details.

User Defined Variable Dependent Mechanical Properties

When performing mechanical or thermo-mechanical analyses with (thermo)elastic, (thermo)elasto-plastic or (thermo)elasto-viscoplastic materials in Digimat-MF, some properties can be defined as dependent over user defined variables. User defined variables are variables

which evolutions must be given by the user and that are not linked to physical quantities like strains, stresses or temperature.

When defining parameter dependencies (see Figure 3-14), User variable (with index 1 to 5) will appear in the Variables column. As user-defined variables evolve independently of other quantities, for every material parameter that dependent over such variable, an associated loading will be added to the analysis. This loading will allow the user to define the evolution of the user defined variable with respect to analysis time.

The workflow to set a strain rate dependency **over a material modulus** (e.g. in the Material Parameters tab) is the same as for temperature dependence: the user should first select the parameter to be scaled, the user-defined variable and the function to use, and then hit **Add** button. Such dependencies can also be enabled, disabled and deleted by clicking on the corresponding buttons. So for every user defined variable dependence, the user has typically to define two functions. First a function f_A that will define the dependence of the selected parameter (for example) with respect to the user variable (for example X_1):

$$\Lambda(X_1) = \Lambda_0 f_A(X_1), \tag{3-5}$$

Next a function f_B that will be associated to the loading and that will define the value of X_1 at every time t:

$$X_1(t) = X_1^{ref} f_B(t), \tag{3-6}$$

As for every user-defined loading, the time evolution can be relative (as defined in this example) using a reference value (X_1^{ref}) or absolute (i.e., assuming X_1^{ref} is set to 1).

The user can combine different user-variable dependences, as well as the temperature dependency, for a single material parameters. Their effects are multiplied such that for example:

$$\Lambda(X_1, X_2, T) = \Lambda_0 f_A(X_1) f_C(X_2) f_D(T), \tag{3-7}$$

To set a user-variable dependency **over a failure parameter moduli** (e.g. in the Failure Indicator tab), the user should first check the box **Use dependent parameters** in the **Failure indicator definition** tab, then click on the **Create** button. This opens the **Dependency** tab, in which a table enables one to define one or several dependencies over each failure parameter. Please refer Setting Dependencies Over Strength Parameters for more information on this subject.

Please note that:

- User defined variables are only available in Digimat-MF
- The naming of the user defined variable is generic but, for example, User variable 1 is assumed to represent the same quantity for all material parameters that define a dependence with respect to that variable. The parameters that can be user-defined variable dependent are currently:
    - Elastic parameters (Young's modulus and Poisson's ration) for the elastic material model

- Elastic parameters (Young's modulus and Poisson's ration) and coefficient of thermal expansion for the thermo-elastic material model
- Plastic parameters (yield stress and hardening parameters) for the elasto-plastic material model
- Plastic parameters (yield stress and hardening parameters) and coefficient of thermal expansion for the thermo-elastoplastic material model
- Viscoplastic parameters (yield stress, hardening and viscosity parameters) for the elasto-viscoplastic material model
- Viscoplastic parameters (yield stress, hardening and viscosity parameters) and coefficient of thermal expansion for the thermo-elastoviscoplastic material model.
- All the strengths parameters defined by the various failure indicators.

N-dimensional Dependent Mechanical Properties

In some very specific configurations, material mechanical properties may depend on several variables. This is for example the case when performing thermo-mechanical analyses with thermo-strain rate elasto-plastic materials in Digimat-MF (see Thermo-strain Rate Elasto-plasticity), some mechanical parameters can depend on both the strain rate and the temperature.

In these cases, the graphical user interface enables to assign N-dimensional dependencies (see Figure 3-17). The procedure to assign such dependencies is very similar to the classical case, except that the user selects several dependency variables, and an N-dimensional piecewise function (see Functions). Obviously, the dimension of the function must match the number of variables selected. The mapping from dependency variables to function X keys, as well as the interpolation type, depends on the material model and the variables type.

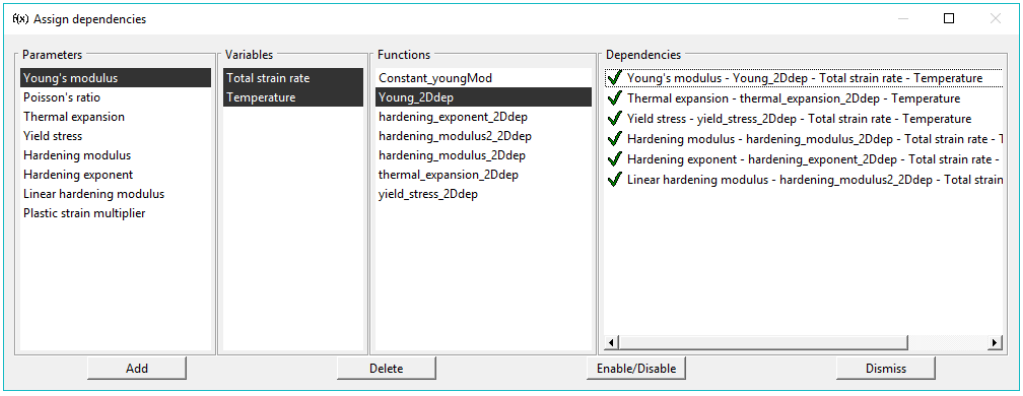


Figure 3-17 Defining N-dimensional dependency for a material parameter.

Another case where N-dimensional dependencies may be involved is Pseudo-grain fatigue with mean stress sensitivity. In this case, the mean stress sensitivity can be described as a 2D function

of the load ratio (R) and number of cycle (N_c). The assignment workflow is slightly different in this case; refer Usage for more information on this topic.

4 Analysis Parameters

- Analyses
- General Parameters
- Integration Parameters

Analyses

It is possible to define several analyses in a single Digimat-MF window. Each analysis being unique, the analysis parameters are defined under each of them. This contains some general parameters, more about the type of analysis it is intended to perform, and several integration parameters.

Most of the parameters are set to default values that work correctly for most general cases. It is nevertheless important to know them well in order to be in a comfortable position to adapt each time it is necessary.

General Parameters

An overview of the **General parameters** tab is provided in Figure 4-1. The different zones of this window are numbered to help you follow up with the description of each of those parameter zones.

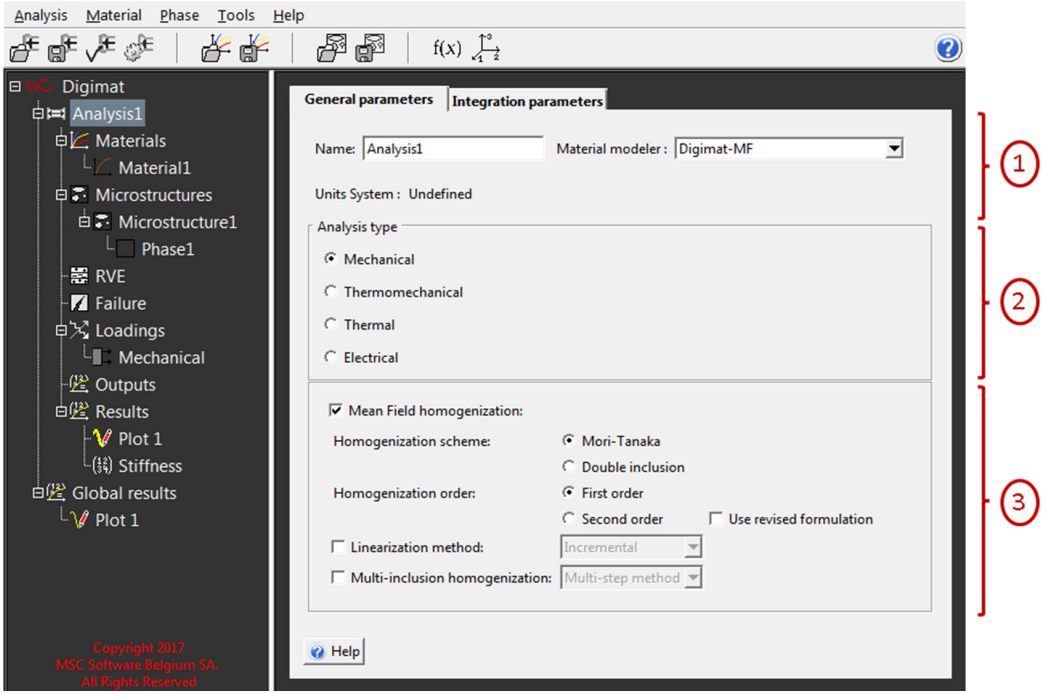


Figure 4-1 General parameters tab containing some analysis parameters.

Name and Material Modeler

Each **Name** must be characterized by a unique name. By default, Digimat sets this field to Analysis and increment the i number to make the name unique. Each analysis must have a different name that does not contain any space nor quotes. Obviously, you can customize the name field to give any name you want to your analysis.

You can use two **material modelers**. By default, Digimat-MF is selected and allows continuing working in Digimat-MF. If wished, it is also possible to transfer the analysis into Digimat-FE to work with it using the FE approach on the RVE. A **Digimat-FE** window opens and are transferred into this window all material, phase and loading information in addition of the analysis parameters.

Analysis Type

There are five types of analysis: mechanical, thermomechanical, thermal, electrical and coupled thermal mechanical. Those analysis types are supported for all loading sources (Digimat-MF stand-alone or interfaced computations). The analysis type determines the proposed material models, the required applied loads and the computed physical properties.

Mechanical

- Such analysis gives access to several material models being: elastic, viscoelastic, hyperelastic, elasto-plastic (with or without damage), elasto-viscoplastic, viscoelastic-viscoplastic, Leonov-EGP.
- It requires the definition of a **mechanical loading** (uniaxial strain, multiaxial strain, etc.) in order to compute the mechanical response (stress-strain curves, stiffness, etc.) of the composite RVE.

Thermo-mechanical

- Such analysis gives access to two material models being: thermo-elastic, thermo-hyperelastic.
- It requires the definition of a **mechanical loading** (multiaxial strain) combined to a thermal loading (temperature) in order to compute the thermo-mechanical response (stress-strain curves, stiffness, thermal expansion) of the composite RVE.

Thermal

- Such analysis gives access to one: the Fourier material model.
- It requires the definition of a **thermal loading** (temperature gradient) in order to compute the thermal response (heat flux, thermal conductivity) of the composite RVE.

Electrical

- Such analysis gives access to one: the Ohm material model.

- It requires the definition of an **electrical loading** (voltage gradient or electric field) in order to compute the electrical response (current density, electrical conductivity) of the composite RVE.

Coupled Thermal Mechanical

- Such analysis enables to combine the Fourier material model with thermomechanical model.
- Two mechanical model are currently supported, a thermoelastic and a thermoviscoelastic one.
- The available loading are only the thermomechanical one, a mechanical and a temperature one. The thermal loading is enforced by default in the solver.
- The results of this analysis are a thermomechanical stress-strain curve as well as stiffness and conductivity at initial and final time.

Mean-field Homogenization

This section about homogenization lists the different homogenization schemes and methods you can use when computing in Digimat. The default parameters are generally the most adapted ones but it is sometimes necessary to modify them. For more information about all homogenization schemes and methods, see Mean-field Homogenization Theory.

Notice there is a check box in front of mean field homogenization. It should be checked on as soon as there is more than one phase defined in the microstructure (that is, composite material). And if you intend to compute only the matrix response (that is, one phase material), then this check box should be checked off.

The two boxes **Linearization method** and **Multi-inclusion homogenization** directly control the behavior of your analysis. If these box lists are not toggle on, Digimat will use the recommended default technique for the analysis. This default depends on the material model and the microstructure. By toggling on the box list, you can also force Digimat to use an individual non-default setting of your own choice (even if it is not the advised technique).

Integration Parameters

Time Intervals

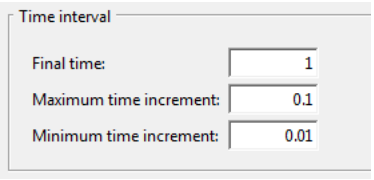


Figure 4-2 Definition of time interval in Digimat-MF analysis.

The time intervals are important parameters to be adapted when working with strain rate dependent materials (i.e., viscoelastic, elasto-viscoplastic, . . .) and when working with user-defined loadings. In those cases, the notion of time in the analysis is very significant and the default parameters are no more guaranteed to work.

The **final time** specifies the total time of the Digimat analysis. The default value is 1s. For time-independent material laws, the absolute value of the final time is not important. If defining user-defined loadings, it is important to update the final time such that the value is the same as the final time of the user-defined loading function. If the update is forgotten, Digimat-MF should nevertheless update it so that it matches that of the user-defined loading function.

The **minimum** and **maximum time increments** are used to set the lower and upper bounds of Digimat's automatic time incrementation. They must be set to a value lower than the **final time**, and the minimum time increment must be set to a value itself lower than the maximum one. When they are set to inadequate values, or simply based on the nonlinear properties of the materials defined in the analysis, Digimat computes its own values and use them during the analysis. The size of the minimum and maximum time increments fixed by Digimat and the actual time steps can all be checked in the `*.log` file generated during the analysis.

Homogenization Scheme Control & Loading Equilibrium Control

In Digimat, you can control the tolerance for two iterative schemes: one for the loading (see Figure 4-4) and other for the homogenization (see Figure 4-3), respectively, "Loop 1" and "Loop 2" in Figure 4-5.

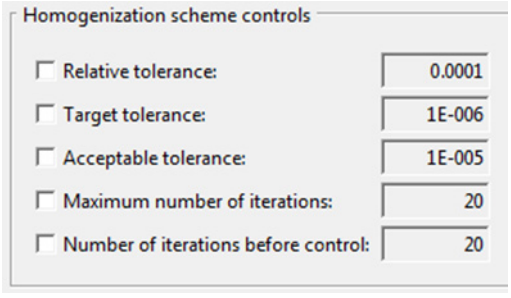


Figure 4-3 Definition of homogenization scheme parameters in Digimat-MF analysis.

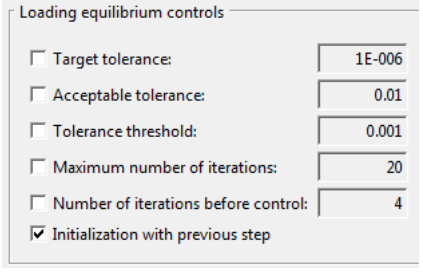


Figure 4-4 Definition of loading scheme parameters in Digimat-MF analysis.

For each, user can define:

- A relative tolerance that drives the stopping criterion
- A target tolerance that set an additional stopping criterion in absolute value
- An acceptable tolerance that sets the maximal level of error so that the iterative process is considered as converged when the maximum number of iterations is reached and that residual is smaller than acceptable tolerance.

The latter is defined in absolute value and is typically one order of magnitude above the target tolerance. Note that loading iterative process stop as soon as either the relative or the absolute tolerance is reached whereas both stopping criterion must be satisfied for the homogenization iterative scheme to stop.

If the acceptable tolerances are too restrictive, it can possibly lead to convergence issues. If they are too permissive, the risk is then that the computations be inaccurate.

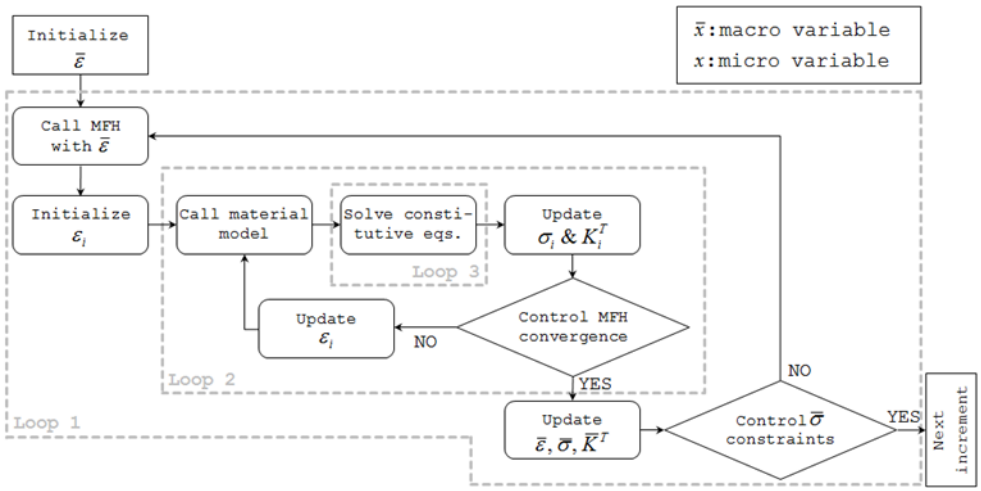


Figure 4-5 Algorithmic iterative process of Digimat.

The iterative computation process performed by Digimat is essentially a two-phase process:

- When the iteration number is below the monitoring iteration number(4 by default), the residual is compared to the target tolerance. The iterative process is said to have converged if the residual is lower than the target tolerance.
- When the iteration number is above the monitoring iteration number, the residual is compared to the acceptable tolerance. If the forecasted number of iterations until convergence is larger than the maximum number of iterations specified in the `.mat` file, convergence is accelerated using another numerical method.

- The default number of iterations is generally well adapted and need to be modified only in some particular cases.

Introduction of a relative homogenization tolerance in Digimat 2016.1 removes the need to use a more restrictive target homogenization tolerance in explicit FE analysis with respect to the one used in implicit FE analysis. Use of a relative tolerance removes the dependency of the stopping criterion to strain amplitude.

Remark 1: In addition to Digimat-MF computations, the loading tolerances settings are only used in simulations involving shell elements. This tolerance is used when verifying the plane stress state. For solid elements this is of no use.

Remark 2: The loading equilibrium iteration starts from an initial guess computed with the previous step, unless the "Initialization with previous step" option is unchecked.

Augmented Lagrangian Scheme Controls

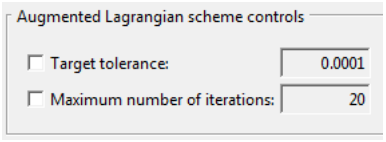


Figure 4-6 Augmented Lagrangian parameters in Digimat-MF analysis.

A **target tolerance** is used in the augmented Lagrangian scheme involved in the incompressibility constraint. This can be applied on hyperelastic materials. The tolerance quantifies the maximum average volume change Digimat can accept for a converged solution.

The **maximum number of iterations controls** the maximum number of iterations allowed in the augmented Lagrangian scheme. If the number of iterations to meet convergence during computation exceeds the value of this parameter, Digimat triggers a time step reduction.

Remark: Default values exist for those **tolerance** and **maximum number of iterations** parameters. In some cases, depending on all the parameters involved in the analysis, Digimat modifies the default values in order to get better results or to avoid uninteresting computations (i.e., computation which doesn't change the results). The effective tolerances and maximum number of iterations used in the computation are written in Digimat log file. To impose a given tolerance or a maximum number of iterations, one has to check the box in front of the parameter under consideration and to change the values appearing in the text fields. If doing so, Digimat will not change the defined values even if they are not best suited to the current analysis.

High Cycle Fatigue Controls

The high cycle fatigue controls are only available for a fatigue analysis, i.e., when a fatigue failure indicator is defined and assigned to the composite. Please report to Fatigue Models and Matrix Damage Fatigue Model for a comprehensive description of the fatigue models implemented in Digimat-MF.

The **dichotomy algorithm tolerance** and **dichotomy algorithm number of iterations** parameters are used by the pseudo grain fatigue model, when the fatigue loading is defined for a range of stress amplitudes (see Fatigue Models).

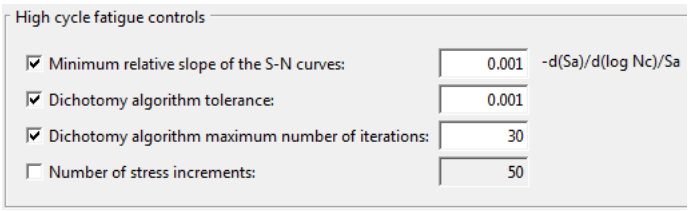


Figure 4-7 High cyclic fatigue parameters in Digimat-MF analysis.

The first one defines the (unitless) target in terms of average failure indicator and the second defines the maximum number of iterations of the algorithm before non-convergence is declared.

The **minimum relative slope of the S-N curves** is a default parameter used for all fatigue models. It triggers an automatic test (and sometimes a correction) on the S-N curves computed by Digimat-MF, to force strictly decreasing values of the stress amplitude, when the critical number of cycles increase. This test comes from physical considerations and can be disabled by defining a negative minimum relative slope.

The **Number of stress increments** parameter is used by the matrix damage fatigue model for the jump-in cycles procedure (see Matrix Damage Fatigue Model). It gives the number of stress increments that are used to evaluate the S-N curve. The larger this parameter, the more precise the results, but also the larger the CPU time!

Remark: This parameter does not affect the number of points in the output files and plots, it only affects the quality of the results.

Integration Scheme

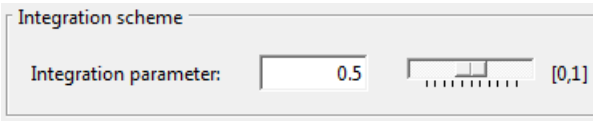


Figure 4-8 Integration scheme parameters in Digimat-MF analysis.

The **integration parameter** is only relevant if mean-field homogenization is set to on and is a real number defining the time integration parameter. It must be larger than zero and smaller or equal to one. The default value of 0.5 corresponds to an implicit mid-point rule time integration scheme. In comparison, when doing explicit computations, 1 is commonly used.

The adaptation of this parameter, depending on whether working in implicit or explicit, is done automatically by Digimat when generating the interface files from Digimat-CAE, based on the code used. But for Digimat-MF, since the solver is implicit, it should be kept to 0.5.

Orientation

Number of angle increments: If the inclusions inside a defined nonlinear composite are oriented with non-fixed orientation (i.e., orientation tensor or random 2D/3D), the orientation space must be discretized. To visualize this discretization step, the angular space can be illustrated by a sphere. This sphere is divided into many segments of equal angular length, with horizontal and vertical lines, as it is done for the planet Earth with the meridians and the parallels.

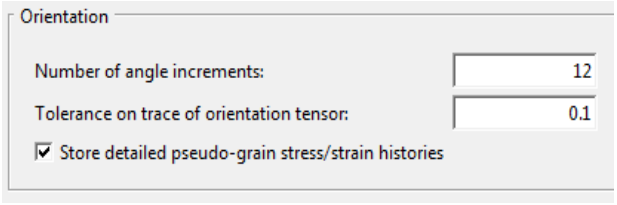


Figure 4-9 Orientation parameters in Digimat-MF analysis.

From their position in the sphere of angular space, all segments describe one unique individual zone of the angular space.

This **number of angle increments** parameter gives the number of angle increments used for the discretization. It must be in the range [6;36] with 12 being the default value. With the use of the Hybrid method, 12 became a good compromise between accuracy, specially for FPGF failure indicators (see First Pseudo-Grain Failure Model) and computation time. The same value is used with and without failure in order to have a smooth transition between material calibrated with and without FPGF failure.

Tolerance on trace of orientation tensor: It defines the tolerance on the orientation tensor trace computed when working with orientation files. One invariant of the orientation tensor stipulates that the trace of the tensor must be equal to 1. If the trace is not equal to one but is within the defined tolerance (the default value is set to 0.1), Digimat automatically corrects the orientation tensor and forces the trace to be equal to 1. If the trace is calculated as outside of the tolerance the orientation tensor is rejected by Digimat and the computation stops.

$$\mathbf{a} = \begin{bmatrix} a_{11} & a_{12} & a_{13} \\ a_{12} & a_{22} & a_{23} \\ a_{13} & a_{23} & a_{33} \end{bmatrix} \tag{4-1}$$

This **a** orientation tensor is corrected such that the trace equals 1, which leads to get the following **a'** tensor:

$$a' = \begin{bmatrix} a_{11}/\,\mathrm{trace}(a) & a_{12} & a_{13} \\ a_{12} & a_{22}/\,\mathrm{trace}(a) & a_{23} \\ a_{13} & a_{23} & a_{33}/\,\mathrm{trace}(a) \end{bmatrix} \quad (4\text{-}2)$$

Remark: We recommend being careful about the orientation tensor trace tolerance. If the trace of the original tensor is too far from 1, it is preferable to check the orientation files to be sure that the original orientation tensors are meaningful!

Store detailed pseudo-grain stress/strain histories: This option enable the storage of the pseudo-grain stress/strain history in order to ensure the best accuracy of the failure criteria. This option is selected by default for all material in order to ensure a direct transition between the material without FPGF failure and the material with FPGF failure.

Collocation Points

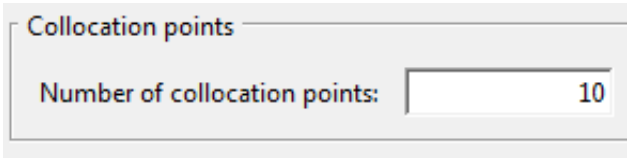


Figure 4-10 Definition of number of collocation points in Digimat-MF analysis.

Number of collocation points: If one of the defined phases of the composite is a viscoelastic material, the user has to define the **number of collocation points**. This number defines the total number of points needed for the numerical inversion of Laplace-Carson transforms. This parameter must be greater than the total number of relaxation time constants of the viscoelastic materials used in the composite. The remaining relaxation time constants will automatically be computed by Digimat. The default value is set to 10.

5 Isotropic Extraction Methods

- Eshelby Approach
- General Method
- Spectral Method
- Modified Spectral Method

Eshelby Approach

In the homogenization process, the Eshelby's tensor is required to compute the strain concentration tensor B^{ε} (see Mean-field Homogenization Theory). It has been shown that good numerical predictions can only be obtained when Eshelby's tensor for nonlinear behavior is computed not from the anisotropic modulus tensor but from an isotropic part of the matrix tangent modulus.

The way to extract an isotropic part from the anisotropic modulus tensor is not unique. Three methods are proposed in Digimat:

- General method
- Spectral method
- Modified spectral one

Their effect on the plastic behavior of the composite is critical (no influence on the elastic behavior will be observed). As a consequence, the choice of the correct method is very important to get good predictions.

Eshelby's tensor is required to compute the strain concentration tensor B^{ε} in the homogenization process,

$$\begin{aligned} B^{\varepsilon} &= \left\{ I + \zeta : C_0^{-1} : [C_1 - C_0] \right\}^{-1} \\ &= \{ I + P : [C_1 - C_0] \}^{-1} \end{aligned} \tag{5-1}$$

where I denotes the fourth-rank identity tensor, ζ: Eshelby's tensor, P: Hill's or polarization tensor, and C_0 and C_1 are the stiffness matrices of the matrix and equivalent inclusion phases.

Eshelby's tensor depends on the inclusion shape and orientation, and the matrix stiffness. In the case of an ellipsoid of revolution and an isotropic stiffness matrix, Eshelby's tensor only depends on the inclusion aspect ratio and orientation, and the matrix Poisson's ratio. For an elasto-(visco)plastic matrix, a tangent Poisson's ratio ν_t is computed from the tangent bulk and shear moduli, K_t and G_t,

$$\nu_t = \frac{3K_t - 2G_t}{2(3K_t + G_t)} \tag{5-2}$$

If the matrix is elasto-(visco)plastic, it can be verified that its tangent modulus tensor is anisotropic, even if the matrix material model is isotropic. An isotropic part of the matrix tangent operator, based on the computation of the tangent shear and bulk moduli is defined as follows:

$$C^{iso} = 3K_t I^{vol} + 2G_t I^{dev} \tag{5-3}$$

where C^{iso} is the isotropic part of the anisotropic modulus tensor, and I^{vol} and I^{dev} are the spherical and deviatoric projection operators, respectively.

Different approaches such as **General, Spectral** and **Modified spectral** methods can be followed to compute K_t and G_t.

General Method

This method can be applied to any material model. It consists in the projection of the anisotropic modulus tensor C^{ani} onto the subspace of isotropic ones, e.g., Bornert et al. (2000). The following relations are then obtained:

$$10G_t = I^{dev} :: C^{ani} \tag{5-4}$$

$$3K_t = I^{vol} :: C^{ani} \tag{5-5}$$

Eshelby's tensor is computed with the tangent bulk and shear moduli.

Spectral Method

For some material models, e.g., J_2-(visco)plasticity, the matrix anisotropic tangent operator is such that a spectral decomposition as proposed by Ponte-Castañeda applies. In the case of J_2-plasticity model, it leads to this specific formulation

$$K_t = K_e \tag{5-6}$$

$$G_t = G_e\left(1 - \frac{3G_e}{3G_e + \frac{dR}{dp}(p)}\right) \tag{5-7}$$

where K_e and G_e denote the elastic bulk and shear moduli, and R and p the isotropic hardening stress and the accumulated plastic strain respectively.

For this method, it is Hill's tensor which is computed with the tangent bulk and shear moduli.

Modified Spectral Method

Modified spectral method is a heuristic generalization of the spectral method,

$$K_t = K_e \tag{5-8}$$

$$G_t = k_G G_e \left(1 - \frac{3 k_t G_e}{3 G_e + \frac{dR}{dp}(k_p p + k_s)} \right) \tag{5-9}$$

where k_G, k_t, k_s and k_p are respectively the global shear multiplier, the plastic shear multiplier, the plastic strain shift and the plastic strain multiplier. Their names are derived from their location in the formula.

It is shown experimentally that the plastic strain multiplier is highly temperature dependent. In fact its value is very different at high temperature in comparison with it at room temperature. Within this framework, it is possible to express the plastic strain multiplier as a function of the temperature in the case of using the modified spectral method.

This feature is available by choosing the **Use dependent parameters** option in the **Parameters** tab of the considered material in the case of using the modified spectral method.

The modified spectral formulation is equivalent to the spectral formulation if you use the following values for the additional parameters $(k_G, k_t, k_s, k_p) = (1, 1, 0, 1)$.

Remarks:

- An important remark is that the isotropic hardening slope $\frac{dR}{dp}$ is not computed with the current value of the accumulated plastic strain, but with another higher value equal to $(k_p p + k_s)$.
- The modified spectral method is designed for a matrix with little hardening reinforced by stiffer fibers, with the purpose of reducing the tangent shear modulus in the "shoulder" region of the stress/strain curve, i.e., the transition from linear elastic behavior to a plateau, or almost, zone. The predicted macroscopic response will be more compliant and closer to experimental composite response than with the regular spectral method.

6 Materials

- Linear (thermo-)elasticity
- Elasto-plasticity: J_2-plasticity Model
- Generalized Drucker-Prager Plasticity
- Hill Plasticity
- Elasto-plastic Damage Material
- Thermo-elastoplasticity
- Elasto-viscoplasticity
- Crystal Plasticity
- Thermo-elasto-viscoplasticity
- Viscoelasticity
- Thermo-viscoelasticity
- Viscoelasticity-viscoplasticity
- Strain Rate Elasto-plasticity
- Thermo-strain Rate Elasto-plasticity
- (Thermo-)hyperelasticity
- Leonov-EGP Model
- Fourier Model - Thermal Conductivity
- Ohm Model - Electrical Conductivity
- Coupled Thermal Mechanical Materials

Linear (thermo-)elasticity

A material is referred to as elastic when its body recovers its original shape once the applied external loadings are removed, i.e., it shows a reversible behavior and no residual strains remain. Both linear (e.g., glass material) and nonlinear (e.g., rubber material) elastic behaviors are observed in laboratories and, as such, both linear and nonlinear models have been developed to represent their behavior.

Although most materials exhibit nonlinear stress/strain relationships when it comes to larger deformations, most of them also exhibit a linear regime for sufficiently small strain. This section focuses on linear elasticity.

First, isothermal elasticity is covered and, second, thermo-elasticity is broached. For nonlinear elasticity, the reader is encouraged to browse the hyperelasticity section of the manual (see (Thermo-)hyperelasticity).

Theory

Linear Elasticity

The linear elasticity law, relating the material strains ε and stresses σ, also referred as Hooke's law, reads

$$\sigma = C : \varepsilon \tag{6-1}$$

where C is called Hooke's operator and the **:** operator denotes the inner product over two indices (i.e., $\sigma_{ij} = C_{ijk}\varepsilon_{kl}$).

Hooke's operator is a fourth-order tensor which, due to symmetries, can be represented by 21 independent scalar components. As a consequence, Hooke's operator can be represented by a 6 $\times$ 6 symmetric matrix, called the stiffness matrix, yielding the matrix form of the linear elasticity law

$$\begin{Bmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{33} \\ \sigma_{12} \\ \sigma_{23} \\ \sigma_{13} \end{Bmatrix} = \begin{bmatrix} C_{11}^{el} & C_{12}^{el} & C_{13}^{el} & C_{14}^{el} & C_{15}^{el} & C_{16}^{el} \\ & C_{22}^{el} & C_{23}^{el} & C_{24}^{el} & C_{25}^{el} & C_{26}^{el} \\ & & C_{33}^{el} & C_{34}^{el} & C_{35}^{el} & C_{36}^{el} \\ & & & C_{44}^{el} & C_{45}^{el} & C_{46}^{el} \\ & & sym & & C_{55}^{el} & C_{56}^{el} \\ & & & & & C_{66}^{el} \end{bmatrix} \begin{Bmatrix} \varepsilon_{11} \\ \varepsilon_{22} \\ \varepsilon_{33} \\ 2\varepsilon_{12} \\ 2\varepsilon_{23} \\ 2\varepsilon_{13} \end{Bmatrix} \tag{6-2}$$

This relation can be inverted to express the strain field components as a function of the stress field components. The inverse of the stiffness matrix is also known as the compliance matrix.

It reads:

$$\begin{Bmatrix} \varepsilon_{11} \\ \varepsilon_{22} \\ \varepsilon_{33} \\ 2\varepsilon_{12} \\ 2\varepsilon_{23} \\ 2\varepsilon_{13} \end{Bmatrix} = \begin{bmatrix} S_{11}^{el} & S_{12}^{el} & S_{13}^{el} & S_{14}^{el} & S_{15}^{el} & S_{16}^{el} \\ & S_{22}^{el} & S_{23}^{el} & S_{24}^{el} & S_{25}^{el} & S_{26}^{el} \\ & & S_{33}^{el} & S_{34}^{el} & S_{35}^{el} & S_{36}^{el} \\ & & & S_{44}^{el} & S_{45}^{el} & S_{46}^{el} \\ & sym & & & S_{55}^{el} & S_{56}^{el} \\ & & & & & S_{66}^{el} \end{bmatrix} \begin{Bmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{33} \\ \sigma_{12} \\ \sigma_{23} \\ \sigma_{13} \end{Bmatrix} \tag{6-3}$$

Depending on the symmetries the material exhibits, the stiffness and compliance matrices degenerate into simpler forms, decreasing the number of independent components needed to describe the material linear elastic behavior.

Note that the shear strains are taken as the engineering shear strains, i.e., Υ_{ij} ij $= 2\varepsilon_{ij}$, for $i \neq j$.

Isotropic Materials

An elastic material is said to be isotropic when its material properties are independent of the considered loading direction. For such a material, the characterization of Hooke's operator requires two independent parameters only, being the isotropic Young's modulus and Poisson's ratio. These parameters are also referred to as engineering constants in the GUI.

For isotropic materials, the compliance matrix degenerates into

$$S^{el} = \frac{1}{E} \begin{bmatrix} 1 & -v & -v & 0 & 0 & 0 \\ & 1 & -v & 0 & 0 & 0 \\ & & 1 & 0 & 0 & 0 \\ & & & 2(1+v) & 0 & 0 \\ & sym & & & 2(1+v) & 0 \\ & & & & & 2(1+v) \end{bmatrix} \tag{6-4}$$

Note that the shear and bulk moduli can be deduced from the Young's modulus and the Poisson's ratio and are defined as follows

$$G = \frac{E}{2(1+v)} \quad and \quad K = \frac{E}{3(1-2v)} \tag{6-5}$$

The Young's modulus is a positive scalar while the Poisson's ratio ranges from -1 to 0.5, the latter bound being synonymous of incompressibility. The GUI will issue an error message if these constraints are not verified when defining an elastic material in Digimat-MF.

Orthotropic Materials

By definition, orthotropic materials present three orthogonal planes within which the material properties are independent of the loading direction. Describing such a material requires nine independent parameters to populate the stiffness (compliance) matrix.

In Digimat-MF, these parameters are taken to be three Young's moduli, three Poisson's ratios and three shear moduli. The compliance matrix for orthotropic materials reads

$$S^{el} = \begin{bmatrix} \frac{1}{E_1} & -\frac{v_{21}}{E_2} & -\frac{v_{31}}{E_3} & 0 & 0 & 0 \\ -\frac{v_{12}}{E_1} & \frac{1}{E_2} & -\frac{v_{32}}{E_3} & 0 & 0 & 0 \\ -\frac{v_{13}}{E_1} & -\frac{v_{23}}{E_2} & -\frac{v_{12}}{E_3} & 0 & 0 & 0 \\ & & & \frac{1}{G_{12}} & 0 & 0 \\ & sym & & & \frac{1}{G_{23}} & 0 \\ & & & & & \frac{1}{G_{13}} \end{bmatrix} \tag{6-6}$$

with, for symmetry reasons, the following equivalence $\nu_{ij}/E_i = \nu_{ji}/E_j$ (no summation). The Poisson's ratio are defined as follows: for a tension test in the 1-direction, $\nu_{12} = -\varepsilon_{22}/\varepsilon_{11}$. Similar definitions hold for the other ratios.

By default, Digimat-MF uses the RVE axis system to define the symmetry planes, but the user is free to modify these by defining a local axis system (see Axis Systems).

Positive definiteness of the compliance (stiffness) matrix requires its components to verify the following constraint equations:

- E1,E2,E3,G12,G13,G23 > 0

- $|12| < (E1/E2)1/2$
- $|13| < (E1/E3)1/2$
- $|23| < (E2/E3)1/2$
- $1 - v_{12}v_{21} - v_{23}v_{32} - v_{13}v_{31} - 2v_{21}v_{32}v_{13} > 0$

Transversely Isotropic Materials

Transversely isotropic materials are a special class of orthotropic materials which present uniform material properties in one plane (e.g., (2,3)-plane) and different properties in the direction normal to this plane (e.g., the 1-axis). To describe such a material, five independent parameters are required. Typical examples of such materials are carbon fibers and a unidirectional composite ply; that is a fiber-reinforced composite where all the fibers are aligned in a fixed given direction.

In Digimat-MF, by default, the symmetry plane is taken as the (2,3)- plane, while the transverse direction corresponds to the 1-axis. Accordingly, the compliance matrix takes the following form

$$S^{el} = \begin{bmatrix} \frac{1}{E_t} & -\frac{v_{pt}}{E_p} & -\frac{v_{pt}}{E_p} & 0 & 0 & 0 \\ -\frac{v_{tp}}{E_t} & \frac{1}{E_p} & -\frac{v_p}{E_p} & 0 & 0 & 0 \\ -\frac{v_{tp}}{E_t} & -\frac{v_p}{E_p} & \frac{1}{E_p} & 0 & 0 & 0 \\ & & & \frac{1}{G_{tp}} & 0 & 0 \\ & sym & & & \frac{2(1+v_p)}{E_p} & 0 \\ & & & & & \frac{1}{G_{tp}} \end{bmatrix} \quad (6\text{-}7)$$

with, as in the orthotropic case, the following equivalence $v_{pt}/E_P = v_{tp}/E_t$. The Young's modulus E_t and E_p are respectively the Axial and In-Plane Young's modulus where t correspond to the principal axis and p correspond to any direction in the plane. The Shear modulus G_{tp} is the transverse shear modulus. If the plane of symmetry is (2, 3), t = 1, p = 2 or 3 and tp = 12 or 13.

Note that the user can select a different coordinate system to define the symmetry planes by creating a local axis system (see Axis Systems) and using it as referential for the material directions.

Several constraints apply to the material parameters:

- $E_p, E_t, G_{tp} > 0$

- $|p| < 1$
- $|pt| < (E_p/E_t)1/2$
- $|tp| < (E_t/E_p)1/2$,
- $1 - v_p^2 - 2v_{tp}v_{pt} - 2v_p v_{tp} v_{pt} > 0$

These constraints ensure the positive definiteness of the compliance (stiffness) matrix.

Anisotropic Materials

Some materials do not present any symmetry at all. They are then referred to as anisotropic materials. The representation of their linear elastic behavior requires the full stiffness (compliance) matrix to be defined, i.e., twenty-one independent parameters are needed to populate the stiffness (compliance) matrix.

In Digimat-MF, by default, the anisotropic stiffness matrix is specified with respect to the RVE 1-2-3 axis system. The user is however free to define it with respect to a different local coordinate system that has been defined prior to the definition of the material.

Curing Law

A curing model is available for isotropic material. The model is described in section Curing Model. When changing state from liquid to rubbery to glassy states, the material stiffness changes drastically. This is done as follows:

- The parameter X_{ref} is an arbitrary reference degree of cure.
- The Young modulus must be defined as a function of temperature $f(T)$ for a degree of cure equal to X_{ref}. This evolution should typically be a decreasing function of temperature, with a sharp drop off at $T = T_g(X_{ref})$, signaling the transition from glassy to rubbery state when the temperature reaches the glass transition temperature.
- The stiffness at any other value of the degree of cure X is $f(T'(X))$, where T' is given by:

$$T' = T_z + \frac{1}{\frac{1}{T - T_z} - \left(\frac{1}{T_g(X) - T_z} - \frac{1}{T_g(X_{ref}) - T_z}\right)} \tag{6-8}$$

where T_z the absolute zero in the temperature scale being used, X_{ref}, the reference degree of cure and T_g, the glass transition temperature. By default, T_z is equal to 0 Kelvin. The advantage of using a thermo-elastic material comparing to a thermo-visco elastic material is that there is no need to identify prony series. The drawback is that it does not take viscous effect into account whereas it is known that they are important during the plateau of cure cycle.

Linear Thermo-elasticity

The theory of linear elasticity briefly introduced in the above sections is valid for isothermal problems, i.e., the sole mechanical loadings are considered and no account of temperature effect on the strain field is taken.

When considering non-isothermal problems, the effect of the temperature field on the material needs to be taken into account.

This can be done by decomposing the strain into two components: (i) a mechanical one, due to external loadings, and (ii) a thermal one, due to a temperature change

$$\varepsilon = \varepsilon^{mec} + \varepsilon^{th} \tag{6-9}$$

Accordingly, the law of linear elasticity can be rewritten as follows

$$\varepsilon = \underbrace{C^{-1}:\sigma}_{e^{mec}} + \varepsilon^{th} \text{ or } \sigma = C:(\varepsilon - \varepsilon^{th}) \tag{6-10}$$

By definition, the Coefficient of Thermal Expansion (CTE, α) relates the thermal strain experienced by a material under a temperature variation. The formulation implemented in Digimat-MF and Digimat-CAE is defined as follow:

$$\varepsilon^{th} = \alpha(T)(T - T_{ref}) - \alpha(T_{init})(T_{init} - T_{ref}) \tag{6-11}$$

T, T_{init} and T_{ref} refer to the current, initial and reference temperatures, the reference temperature being the temperature at which the thermal strain is null. Section Reverse Engineering of CTE explains how to make the reverse engineering of the matrix CTE.

The α tensor, is symmetric and takes various forms depending on the material symmetries. These are reported in Table 6-1.

Note that, for thermo-mechanical analyses in Digimat-MF, temperature-dependent parameters can be defined using scaling functions (see Functions).

Table 6-1 CTE matrices depending on the material symmetry

Material symmetry	Number of independent components	Matrix form
Isotropic	1	$\begin{bmatrix} \alpha & 0 & 0 \\ & \alpha & 0 \\ sym & & \alpha \end{bmatrix}$
Transversely isotropic	2	$\begin{bmatrix} \alpha_t & 0 & 0 \\ & \alpha_p & 0 \\ sym & & \alpha_p \end{bmatrix}$
Orthotropic	3	$\begin{bmatrix} \alpha_1 & 0 & 0 \\ & \alpha_2 & 0 \\ sym & & \alpha_3 \end{bmatrix}$
Anisotropic	6	$\begin{bmatrix} \alpha_{11} & \alpha_{12} & \alpha_{13} \\ & \alpha_{22} & \alpha_{23} \\ sym & & \alpha_{33} \end{bmatrix}$

Units

Digimat-MF is not bound to a particular system of units. Nevertheless, units must be consistent throughout the definition of the material model, e.g., if the Young modulus is given in MPa, so should be the hardening moduli and the yield stress, and similarly for other dimensional parameters of the model. Table 6-2 lists all the parameters introduced in this page, their denomination as well as their corresponding dimensions.

In addition, if the created material is to be used in a coupled simulation with a finite element code, the user will make sure that the chosen unit system are consistent with the one used in the finite element model.

Table 6-2 Parameter names, symbols, dimensions and SI units

Name	Symbol	Dimensions - M, L, T, θ, I	SI units
Density	ρ	ML^{-3}	kg/m^3
Specific heat	c	$L^2T^{-2}\theta^{-1}$	J/kg.K
Thermal conductivity coefficient	k^{th}	$MLT^{-3}\theta^{-1}$	W/m.K

Table 6-2 Parameter names, symbols, dimensions and SI units

Name	Symbol	Dimensions - M, L, T, θ, I	SI units
Thermal flux	Q	MT^{-3}	W/m^2
Temperature	T	θ	K
Volumetric heat supply	R	$ML^{-1}T^{-3}$	W/m^3

Elasto-plasticity: J_2-plasticity Model

Figure 6-1 shows an idealized stress-strain response of a polymer under uniaxial tension in the x-direction. It can be seen that this material exhibits nonlinear behavior as soon as the stress exceeds a threshold value called yield stress, σ_Y. If the specimen is unloaded at any point along A-B, a permanent deformation ε_p, or plastic deformation, is observed. If the stress/strain response is independent of the strain rate, then this material can be modeled using elasto-plasticity theory. If a dependence on the strain rate of the stress/strain response is observed, models based on elasto-viscoplasticity or viscoelasticity-viscoplasticity should be used, see the respective sections.

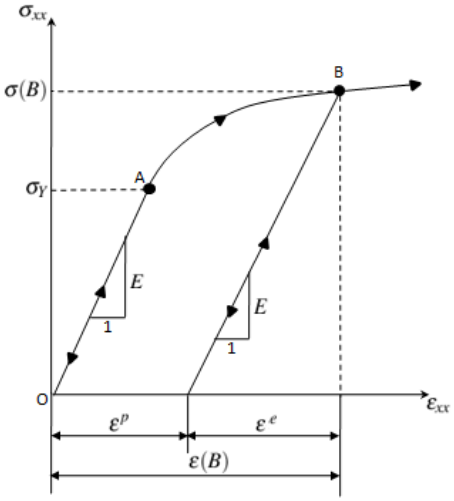


Figure 6-1 Idealized stress/strain response of a polymer under uniaxial tension in the x–direction.

Theory

The elasto-plastic (EP) constitutive model available in Digimat is the J_2-plasticity model. This model is based on the von Mises equivalent stress σ_{eq}, defined as:

$$\sigma_{eq} = \sqrt{3J_2(\sigma)} = \left(\frac{3}{2}S{:}S\right)^{1/2} \tag{6-12}$$

where, $J_2(\sigma)$ is the second invariant of the deviatoric stress tensor S, and is expressed as:

$$\begin{aligned} J_2(\sigma) &= \frac{1}{2}(S:S) \\ &= \frac{1}{2}\left[\left(\sigma - \frac{1}{3}T_r(\sigma)I\right) : \left(\sigma - \frac{1}{3}T_r(\sigma)I\right)\right] \\ &= \frac{1}{6}[(\sigma_{11}-\sigma_{22})^2 + (\sigma_{22}-\sigma_{33})^2 + (\sigma_{33}-\sigma_{11})^2] + 3[\sigma_{12}^2 + \sigma_{23}^2 + \sigma_{31}^2] \end{aligned} \tag{6-13}$$

Remark: For uniaxial loadings, the von Mises equivalent stress is equal to the axial stress.

In this constitutive model, the response is assumed to be linear elastic as long as the following condition is satisfied:

$$\sigma_{eq} \leq \sigma_Y \tag{6-14}$$

where σ_Y is a material parameter known as the initial yield stress.

The total strain observed by the material is assumed to be the sum of the plastic strain and elastic strain,

$$\varepsilon = \varepsilon^e + \varepsilon^p \tag{6-15}$$

The Cauchy stress and the elastic strain are then related by,

$$\sigma = C : \varepsilon^e \tag{6-16}$$

where C is Hooke's operator. When σ_{eq} exceeds the initial yield stress, the response becomes nonlinear and plastic deformation appears. In this case, the Cauchy stress is given by,

$$\sigma_{eq} = \sigma_Y + R(p) \tag{6-17}$$

where $R(p)$ is the hardening stress and p the accumulated plastic strain is expressed as,

$$p(t) = \int_0^t \dot{p}(\tau)d\tau \tag{6-18}$$

with

$$\dot{p} = \frac{2}{3}\sqrt{J_2(\dot{\varepsilon}^p)} = \sqrt{\frac{2}{3}\dot{\varepsilon}^p : \dot{\varepsilon}^p} \tag{6-19}$$

Remark: For uniaxial loadings, the accumulated plastic strain is equal to the axial plastic strain. The 2/3 factor enables to account for the transversal shrinkage due to the incompressibility of plastic strains.

A yield function f (σ, R) can be defined,

$$f(\sigma, R) = \sigma_{eq} - \sigma_Y - R(p) \leq 0 \tag{6-20}$$

If f (σ, R) is lower than zero, the material evolves in the elastic domain. Otherwise it is in the plastic region.

The evolution of the plastic strain tensor ε^p is given by the normality rule,

$$\dot{\varepsilon}^p = \dot{p}\frac{\partial f}{\partial \sigma} \tag{6-21}$$

Isotropic Hardening Model

Five laws are available for the hardening stress:

- Power law

  $$R(p) = kp^m \tag{6-22}$$

- Exponential law

  $$R(p) = R_\infty[1 - \exp(-mp)] \tag{6-23}$$

- Exponential and linear law

  $$R(p) = kp + R_\infty[1 - \exp(-mp)] \tag{6-24}$$

- Swift law

  $$R(p) = k(p + p_0)^m \tag{6-25}$$

- Piecewise law

  Defined by a function. This monotonically increasing function describes the evolution of the yield stress as a function of the accumulated plastic strain, with at least 4 points. The first point of the function represents the initial yield stress $(p = 0)$.

If the tensile stress/strain curve of the material exhibits a horizontal plateau, the exponential hardening law should be retained. The exponential and linear law is used when a plateau is almost reached but the stress level keeps increasing slowly. The Swift law is commonly used for prestrained metallic materials with $\sigma_Y = 0$.

Kinematic Hardening Model

The kinematic hardening model should be used to predict the EP behavior of materials (like metals or polymer) under cyclic loadings. Consider the strain history of Figure 6-2. It leads to the idealized stress/strain response shown in Figure 6-3.

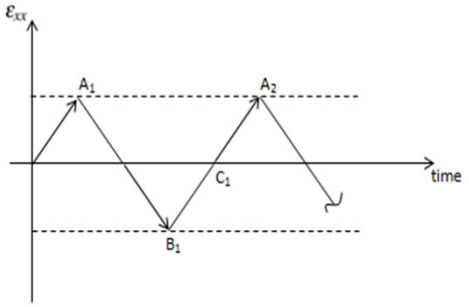


Figure 6-2 Strain history during cyclic loading.

In this case, the yield surface size is equal to $2(\sigma_y + R)$ but its center is no longer located on the x-axis.

In other words, it is seen that the yield stresses in compression and in tension are no longer equal to each other, and the compressive yield stress, in absolute value, is smaller than the tensile one. This is known as the Baushinger effect. This effect cannot be predicted by an isotropic hardening model alone and requires the use of a kinematic hardening model. In Digimat, the Armstrong-Frederick-Chaboche model with combined nonlinear isotropic and kinematic hardening is available.

The general EP model becomes:

$$\sigma = C : (\varepsilon - \varepsilon^p) \tag{6-26}$$

$$f = \sqrt{J_2(\sigma - X)} - R(p) - \sigma_Y \tag{6-27}$$

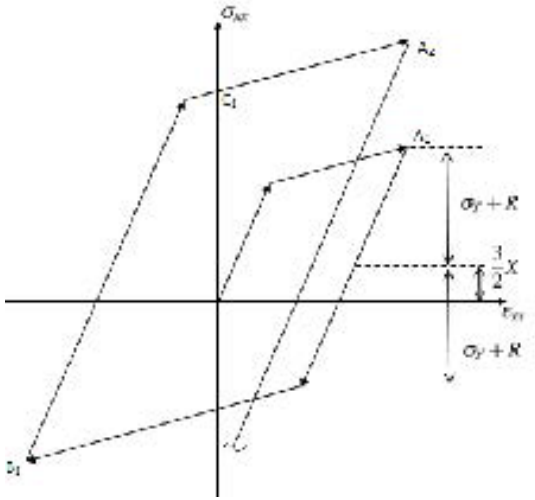


Figure 6-3 Idealized stress-strain response of a polymer under cyclic loading

Two kinematic hardening models are available.

- Linear law

$$\dot{X} = a\dot{\varepsilon}^{p} \tag{6-28}$$

- Linear and restoration law

$$\dot{X} = a\dot{\varepsilon}^{p} - bX\dot{p} \tag{6-29}$$

where the variable X, the so-called back stress, measures the translation of the center of the yield surface, a and b are the linear hardening modulus and restoration parameters of the kinematic hardening model. The overhead dot denotes the time derivative operator.

Isotropic Extraction Method

In the homogenization process of EP materials, the isotropic part of the tangent stiffness tensor must be extracted. Three different methods are available in Digimat. They are described in detail in Chapter 5, Isotropic Extraction Methods.

Units

Digimat-MF is not bound to a particular system of units. Nevertheless, units must be consistent throughout the definition of the material model, e.g., if the Young modulus is given in MPa, so should be the hardening moduli and the yield stress, and similarly for other dimensional parameters of the model.

Table 6-3 lists all the parameters introduced in this page, their denomination as well as their corresponding dimensions.

In addition, if the created material is to be used in a coupled simulation with a finite element code, the user will make sure that the chosen unit system is consistent with the one used in the finite element model.

Table 6-3 Parameter names, symbols, dimensions and SI units

Name	Symbol	Dimensions M, L, T, θ , I	SI Unit
Young's modulus	E	$ML^{-1}T^{-2}$	$Pa = N/m^2$
Poisson's ratio	ν	1	-
Yield stress	σ_Y	$ML^{-1}T^{-2}$	$Pa = N/m^2$
Linear hardening modulus	k	$ML^{-1}T^{-2}$	$Pa = N/m^2$
Hardening modulus	R_∞	$ML^{-1}T^{-2}$	$Pa = N/m^2$
Hardening exponent	m	1	-
Prestrain	p_0	1	-

Table 6-3 Parameter names, symbols, dimensions and SI units

Name	Symbol	Dimensions M, L, T, θ , I	SI Unit
Linear hardening modulus	a	$ML^{-1}T^{-2}$	$Pa = N/m^2$
Restoration	b	1	-

Generalized Drucker-Prager Plasticity

The generalized Drucker-Prager plasticity model is an extension of the original model proposed by Drucker and Prager (1952). This plasticity model, contrarily to the classical J_2-plasticity model, is pressure-dependent. This means that it takes into account pressure effects in the plastic regime. It is generally used to describe the behavior of materials which are pressure-dependent such as polymers and rocks.

Theory

The generalized Drucker-Prager model is defined by a yield surface and a flow potential which depend on the von Mises stress and the hydrostatic pressure. The yield surface is defined as follows

$$\Phi(\sigma_{eq}, \sigma_m, p) = M_\Phi\left(\frac{\sigma_{eq}}{\sigma_y}\right)^q - H_\Phi\sigma_m - \sigma_t(p) \tag{6-30}$$

where $M_\phi > 0$, $q > 0$ and $0 \le H_\phi \le 1$ respectively are the yield stress coefficient, the yield stress exponent and the yield pressure coefficient. These parameters are independent of the plastic deformation. Contrarily to other plasticity models implemented in Digimat-MF, this particular definition uses both the hydrostatic pressure:

$$\sigma_m = -\frac{1}{3}I_1(\sigma) = -\frac{1}{3}(\sigma_{11} + \sigma_{22} + \sigma_{33}) \tag{6-31}$$

and the equivalent von Mises stress:

$$\sigma_{eq} = \sqrt{3J_2(\sigma)} = \left(\frac{1}{2}[(\sigma_{11} - \sigma_{22})^2 + (\sigma_{22} - \sigma_{33})^2 + (\sigma_{33} - \sigma_{11})^2] + 3[\sigma_{12}^2 + \sigma_{23}^2 + \sigma_{31}^2]\right)^{1/2} \tag{6-32}$$

Three different types of hardening functions $\sigma_t(p)$ can be considered. These functions are related to the type of test that was performed to calibrate the Drucker-Prager model. Once selected in Digimat-MF, the hardening function will be used throughout the calculations. In the following expressions, the overhead bar refers to the current von Mises yield stress, i.e.,

$$\bar{\sigma}(p) = \sigma_y + R(p) \tag{6-33}$$

where σ_y and $R(p)$ respectively denote the initial von Mises yield stress and the isotropic hardening.

- Tensile test:

$$\sigma_t(p) = M_\Phi\left(\frac{\bar{\sigma}(p)}{\sigma_y}\right)^q + H_\phi\frac{\bar{\sigma}(p)}{3} \tag{6-34}$$

- Compression test:

$$\sigma_t(p) = M_\Phi\left(\frac{\bar{\sigma}(p)}{\sigma_y}\right)^q - H_\Phi\frac{\bar{\sigma}(p)}{3} \tag{6-35}$$

- Shear test:

$$\sigma_t(p) = M_\phi\left(\frac{\bar{\sigma}(p)}{\sigma_y}\right)^q \tag{6-36}$$

The yield stress exponent is assumed to be equal to 1 by default. A schematic representation of the yield surface in the (σ_{eq}, σ_m)-plane is presented in Figure 6-4. The yield surface intersects the hydrostatic pressure and von Mises axes at points B and A such that

$$B = \left(\sigma_m = -\frac{\sigma_t(p)}{H_\phi}, \sigma_{eq} = 0\right) \text{ and } A = \left(\sigma_m = 0, \sigma_{eq} = \sigma_y\left[\frac{\sigma_t(p)}{M_\Phi}\right]^{1/q}\right) \tag{6-37}$$

The flow potential is defined as follows:

$$G(\sigma_{eq}, \sigma_m) = \sqrt{[\xi\sigma_y\tan(\phi)]^2 + \sigma_{eq}^2} - \sigma_m\tan(\phi) \tag{6-38}$$

where $\xi > 0$ denotes the eccentricity, σ_y the initial yield stress of the material and ϕ the dilatation angle.

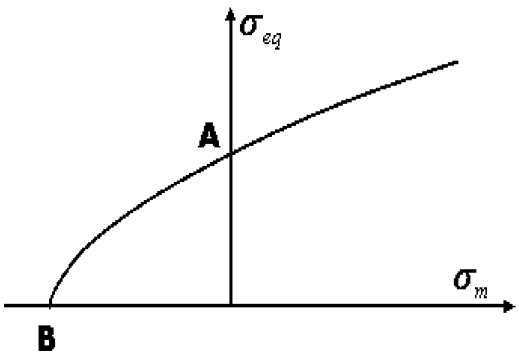


Figure 6-4 Schematic representation of the yield surface in the (σ_{eq}, σ_m)-plane.

The flow potential is also defined through a nonlinear relation between the first two invariants of the Cauchy stress tensor σ_m and σ_{eq}. This relation becomes linear and independent of the dilatation angle if the eccentricity tends to zero. The eccentricity has no strong effect on the response of the material but improves the stability of the plasticity algorithm. The default value of the eccentricity is 0.1 while the dilation angle is set to zero by default.

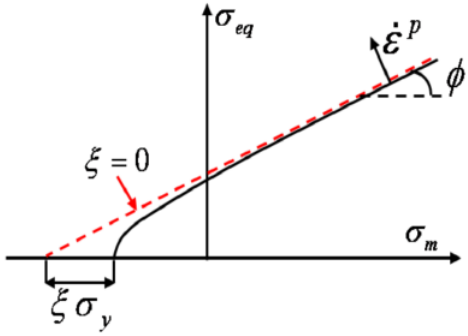


Figure 6-5 Schematic representation of the flow surface in the (σ_{eq}, σ_m)-plane

This general definition of the yield surface and the flow potential implies that the flow rule is non-associated, i.e., the plastic flow is normal to the flow potential as shown in Figure 6-5. The evolution of the plastic strain is governed by the plastic flow rule:

$$\dot{\varepsilon}^p = \dot{\lambda}\frac{\partial G}{\partial \sigma}(\sigma_{eq}, \sigma_m) \tag{6-39}$$

where the overhead dot denotes the time derivative and λ the plastic multiplier. From this definition of the plastic strain evolution and the general definition of the plastic flow rule, it is straightforward to show that the plastic strain is not only deviatoric, as it is the case in the classical J_2-plasticity theory, but is also dependent on a volumetric part, function of the hydrostatic pressure. The accumulated plastic strain rate is assumed to vary according to the equivalent plastic work

$$\bar{\sigma}(p)\dot{p} = \sigma : \dot{\varepsilon}^p \tag{6-40}$$

Let us note that the use of the following particular set of parameters leads to recover the associated J_2 plasticity model as a special case $M_\phi = 1$, $q = 1$, $H_\phi = 0$ and $\phi = 0$.

Units

Digimat-MF is not bound to a particular system of units. Nevertheless, units must be consistent throughout the definition of the material model, e.g., if the Young modulus is given in MPa, so should be the hardening moduli and the yield stress, and similarly for other dimensional parameters of the model.

Table 6-4 lists all the parameters introduced in this page, their denomination as well as their corresponding dimensions.

In addition, if the created material is to be used in a coupled simulation with a finite element code, the user will make sure that the chosen unit system is consistent with the one used in the finite element model.

Table 6-4 Parameter names, symbols, dimensions and SI units (except for dilatation angle)

Name	Symbol	Dimensions - M, L, T, θ, I	SI units
Young's modulus	E	$ML^{-1}T^{-2}$	$Pa = N/m^2$
Poisson's ratio	υ	1	-
Yield Stress	σ_Y	$ML^{-1}T^{-2}$	$Pa = N/m^2$
Hardening modulus	k	$ML^{-1}T^{-2}$	$Pa = N/m^2$
Hardening modulus	R_∞	$ML^{-1}T^{-2}$	$Pa = N/m^2$
Hardening exponent	m	1	-
Eccentricity	ξ	1	-
Dilatation angle	Φ	1	degree
Yield stress coefficient	M_Φ	$ML^{-1}T^{-2}$	$Pa = N/m^2$
Yield stress exponent	q	1	-
Yield pressure coefficient	H_Φ	1	-

Hill Plasticity

Anisotropic plastic deformation is modelled by the Hill yield criterion developed by Rodney Hill (1948). This quadratic yield locus is a straightforward extension of the von Mises yield locus, through the use of yield stress ratios β_{ij}.

Theory

The equivalent yield stress is written as

$$2\sigma_{Hill}^2 = F(\sigma_{22}-\sigma_{33})^2 + G(\sigma_{33}-\sigma_{11})^2 + H(\sigma_{11}-\sigma_{22})^2 + 6L\sigma_{31}^2 + 6M\sigma_{23}^2 + 6N\sigma_{12}^2 \quad (6\text{-}41)$$

where *F*, *G*, *H*, *L*, *M* and *N* are constants defined as:

$$F = \frac{1}{\beta_{22}} + \frac{1}{\beta_{33}} - \frac{1}{\beta_{11}} \quad (6\text{-}42)$$

$$G = \frac{1}{\beta_{33}} + \frac{1}{\beta_{11}} - \frac{1}{\beta_{22}} \quad (6\text{-}43)$$

$$H = \frac{1}{\beta_{11}} + \frac{1}{\beta_{22}} - \frac{1}{\beta_{33}} \quad (6\text{-}44)$$

$$L = \frac{1}{\beta_{23}} \quad (6\text{-}45)$$

$$M = \frac{1}{\beta_{31}} \quad (6\text{-}46)$$

$$N = \frac{1}{\beta_{12}} \quad (6\text{-}47)$$

The yield ratios β_{ij} are defined with respect to a reference yield stress, σ_0 (given by $\sigma_Y + R(p=0)$), such that if σ_{ij} is applied as the only non-zero stress, the corresponding yield stress $\bar{\sigma}_{ij}$ is $\beta_{ij}\sigma_0$. The yield stress ratios are obtained by tests of the material in different orientations. For shear components, the ratio of the reference shear yield ($\sigma_0/\sqrt{3}$) to the actual shear yield is used.

The yield function is defined as:

$$f(\sigma, p) = \sigma_{Hill} - \sigma_Y - R(p) \quad (6\text{-}48)$$

while the normality rule developed for J_2-plasticity still holds.

Defining Hill Plasticity on the Basis of Lankford's r-values

Hill's (1948) Yield Criterion has been extensively used in sheet metal forming, especially for steel. In metal stamping applications, it is common to find the anisotropic experimental material data given in terms of the so-called Lankford's coefficients (r-values), defined as the ratios of width strain to thickness strain. These experimental data can be related to the Digimat input for the Hill's Yield Criterion for plane stress case as given below.

Testing material samples in the three directions $\theta = 0°$ (Rolling direction), $45°$ and $90°$ (Transverse direction) allows to determine σ_0, σ_{90}, r_0, r_{45} and r_{90}. The yield stress in the normal direction (N) can be written as:

$$\sigma_N = \sigma_0 \sqrt{\frac{r_{90}(1 + r_0)}{r_0 + r_{90}}} = \sigma_{90} \sqrt{\frac{r_0(1 + r_{90})}{r_0 + r_{90}}} \tag{6-49}$$

The yield stress coefficients are now:

$$\beta_{11} = \frac{\sigma_0}{\sigma_{av}} \tag{6-50}$$

$$\beta_{22} = \frac{\sigma_{90}}{\sigma_{av}} \tag{6-51}$$

$$\beta_{33} = \frac{\sigma_N}{\sigma_{av}} \tag{6-52}$$

$$\beta_{12} = \beta_{33} = \sqrt{\frac{3}{2r_{45} + 1}} \tag{6-53}$$

$$\beta_{23} = \beta_{31} = 1 \tag{6-54}$$

where σ_{av} is the initial yield stress. If the stress-strain curve is averaged from all directions, σ_{av} is defined as $\frac{\sigma_0 + 2\sigma_{45} + \sigma_{90}}{4}$ in orthotropic plasticity.

Units

Digimat-MF is not bound to a particular system of units. Nevertheless, units must be consistent throughout the definition of the material model, e.g., if the Young's modulus is given in MPa, so should be the hardening moduli and the yield stress. And similarly for other dimensional parameters of the model.

Table 6-5 lists all the parameters introduced in this page, their denomination as well as their corresponding dimensions.

In addition, if the created material is to be used in a coupled simulation with a finite element code, the user will make sure that the chosen unit system is consistent with the one used in the finite element model.

Table 6-5 Parameter names, symbols, dimensions and SI units

Name	Symbol	Dimensions M, L, T, θ , I	SI Unit
Young's modulus	E	$ML^{-1}T^{-2}$	$Pa = N/m^2$
Poisson's ratio	ν	1	-
Yield stress	σ_Y	$ML^{-1}T^{-2}$	$Pa = N/m^2$
Linear hardening modulus	k	$ML^{-1}T^{-2}$	$Pa = N/m^2$
Hardening modulus	R_∞	$ML^{-1}T^{-2}$	$Pa = N/m^2$
Hardening exponent	m	1	-
Prestrain	p_0	1	-
Yield stress ratio	β_{ij}	1	-

Elasto-plastic Damage Material

Figure 6-6 shows an idealized stress-strain response of a polymer under uniaxial tension (x-direction) that exhibited progressive damage. The damage leads to a decrease of the stiffness of the material. The damage level can be evaluated by using the following formula

$$D = 1 - \frac{E_D}{E} \tag{6-55}$$

where E and E_D are the Young's moduli of the sane and damaged material. D ranges between 0 and 1. Elasto-plastic materials that do not experience damage typically show a monotonic stress/strain response, i.e., to any strain increase corresponds a positive stress increment. When damage occurs, a reduction of the stiffness is observed as well as, potentially, a change of the stress/strain curve slope sign, i.e., a positive strain increment leads to a negative stress increment.

The EP damage model available in Digimat is the Lemaitre-Chaboche model (1990). It assumes isotropic and ductile damage.

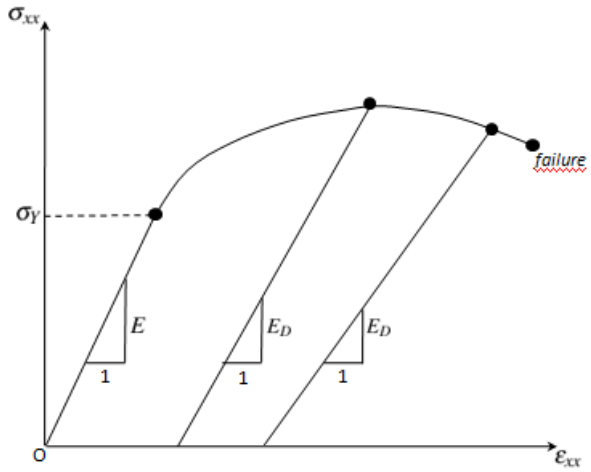


Figure 6-6 Idealized stress/strain response of a polymer under uniaxial tension (x–direction).

J_2-plasticity Damage Model

In Digimat, the Lemaitre-Chaboche damage model is coupled to J_2-plasticity with isotropic hardening. In small strain elasto-plasticity, the decomposition of the total strain into elastic and plastic components reads

$$\varepsilon = \varepsilon^e + \varepsilon^p \tag{6-56}$$

The Lemaitre-Chaboche model considers that damage occurs only in the plastic regime, i.e., when ε^p is positive.

The evolution of the damage is governed by thermodynamics. Let the following variables be introduced:

- σ : the stress tensor
- R: the isotropic hardening stress
- -Y: the force related to the damage.

Let also the following internal variables be defined:

- ε^p : the plastic strain tensor
- r: a scalar variable that represents isotropic hardening
- D: a scalar parameter that represents the damage.

Thermodynamic forces and internal variables are linked together via the specific free energy ψ

$$\rho\psi(\varepsilon^e, r, D) = \frac{1}{2}\varepsilon^e : (1-D)E_0 : \varepsilon^e + \int_0^r R(\zeta)d\zeta \tag{6-57}$$

where E_0 is Hooke's operator and the following state equations

$$\sigma = \rho\frac{\partial\psi}{\partial\varepsilon^e}, \quad A = \rho\frac{\partial\psi}{\partial V} \tag{6-58}$$

where V = (r, D) is a set of internal variables and A = A(R,-Y) is the set of conjugate (dual) thermodynamic forces.

Using the definition of the specific free energy and the above state equations, the following relations can be derived

$$\sigma = (1-D)E_0 : (\varepsilon - \varepsilon^p), \quad R = R(r), \quad Y = \frac{1}{2}\varepsilon^e : E_0 : \varepsilon^e \tag{6-59}$$

Considering damage, the J_2-plasticity yield surface reads

$$f(\sigma, R, D) = \bar{\sigma}_{eq} - \sigma_y - R(r) \leq 0 \tag{6-60}$$

where

$$\bar{\sigma} = \frac{\sigma}{1-D} \tag{6-61}$$

is called effective stress and isotropic hardening is taken into account.

To define the effective stress, let us consider a RVE under a tension force F in the x-direction (see Figure 6-7). Let us denote by A the total cross section area, which can be seen as the sum of a damaged area A_D and a neat resisting area $\tilde{A}$ such that

$$A = A_D + \tilde{A} \tag{6-62}$$

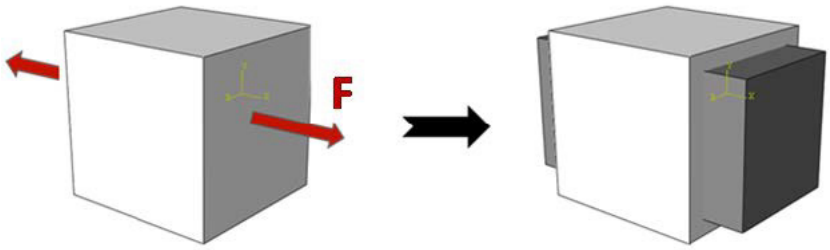


Figure 6-7 Definition of the tensile force applied on the RVE, in the x-direction.

A_D is the surface area of the projection of the flaws due to damage (micro-voids and micro-cavities) on the cross section. Damage variable D is defined as the ratio

$$D = \frac{A_D}{A} \tag{6-63}$$

and ranges between 0 for the sane material and 1 for the fully broken one. Consequently the resisting or effective cross section area is

$$\tilde{A} = (1 - D)A \tag{6-64}$$

By definition of the Cauchy stress σ_{xx} the total force F is

$$F = \sigma_{xx}A \tag{6-65}$$

Similarly, a so-called effective stress is defined such that

$$F = \tilde{\sigma}_{xx}\tilde{A} \tag{6-66}$$

Combining the equations, it is seen that

$$\tilde{\sigma}_{xx} = \frac{\sigma_{xx}}{(1 - D)} \tag{6-67}$$

This represents the fact that the total load F is actually supported by the effective cross section of area Ã. This concept is generalized to a multi-axial stress state as follows. On any material plane of normal $n^*=(n_i)$, we can write, using similar notations to the previous uniaxial case:

$$F_i = (\sigma_{ji}n_j)dA = (\tilde{\sigma}_{ji}n_j)d\tilde{A} \tag{6-68}$$

Noting D(n*) the damage measure on the plane of normal n*, we obtain successively:

$$D(n^*) = \frac{dA - d\tilde{A}}{dA}, \tag{6-69}$$

$$d\tilde{A} = (1 - D(n^*))dA, \tag{6-70}$$

$$\tilde{\sigma}_{ij} = \frac{\sigma_{ij}}{(1 - D(n^*))} \tag{6-71}$$

So it is seen that the effective stress is related to the Cauchy stress via the damage variable in a relation similar to the uniaxial case, and has the same interpretation as before.

One important remark is that the damage $D(n^*)$ should depend on the normal n^*, and is therefore anisotropic. However, in the present model, isotropic damage is assumed and represented by a scalar D which is independent of the orientation n^*.

This effective stress being defined, let us note that when damage occurs, the following relation exists between p and r,

$$\dot{r} = (1 - D)\dot{p} \tag{6-72}$$

The damage evolution is given by the general formula,

$$\dot{D} = \dot{r}\frac{\partial F_D}{\partial y} \tag{6-73}$$

In the Lemaitre-Chaboche model, FD reads

$$F_D(Y, D) = \frac{Y^2}{2S_0}\frac{1}{1-D'} \tag{6-74}$$

where $1/S_0$ is called the damage rate factor, or DRF. The damage rate factor is constant; hence the evolution of D can be rewritten as

$$\dot{D} = \frac{Y}{S_0}\dot{p} \tag{6-75}$$

In order to fully control the damage of the material, two additional parameters are introduced:

- D_c, the critical damage. The material is assumed to fail as soon as D reaches D_c. This leads to the termination of the Digimat computation. D_c can be set smaller than 1. By default, its value is 1.
- pD, the damage initialization threshold, which controls the minimum accumulated plastic strain, p, that must be incurred by the material to start damaging. By default, pD is equal to 0.

Finally, an expanded version of the typical Lemaitre-Chaboche is proposed in Digimat, leading to the following evolution law of D

$$\dot{D} = \frac{\sigma_Y}{S_0}\left(\frac{Y(\bar{\sigma})}{\sigma_Y}\right)^n \dot{p} \tag{6-76}$$

where n is the damage exponent that controls the convexity of $D(p)$.Y denotes the strain energy release rate and has the following expression

$$Y = \frac{1}{2E_0}\left[\frac{\sigma_{eq}}{1-D}\right]^2 R_v, \text{ with } R_v = \frac{2}{3}(1+v) + 3(1-2v)\left[\frac{\sigma_H}{\sigma_{eq}}\right]^2, \tag{6-77}$$

where $\sigma_H = -\frac{1}{3}T_r(\sigma)$ is the hydrostatic stress and $\sigma_{eq} = \sqrt{3J_2\sigma}$ is the von Mises equivalent stress. Damage evolution is therefore dependent on the triaxiality ratio σ_H/σ_{eq} which is known to play a major role in ductile failure.

In the Lemaitre-Chaboche model, ductile damage:

- directly affects the material elastic stiffness and can be experimentally measured by unloading and reloading tests as depicted in Figure 6-6;.

- is coupled to the field function (f) but the latter remains of J_2 form (deviatoric), there is no influence of the hydrostatic pressure (σ_H) on the field.

These two features may be adequate for some materials, e.g., some metals, and inadequate for others. Only sufficient experimental evidence can give guidance on the usefulness of the Lemaitre-Chaboche ductile damage model.

Remark: In the typical J_2-plasticity model without damage, the effective stress is equal to the equivalent stress and the yield surface is expressed as,

$$f(\sigma, R) = \sigma_{eq} - \sigma_Y - R(p) \tag{6-78}$$

Units

Digimat-MF is not bound to a particular system of units. Nevertheless, units must be consistent throughout the definition of the material model, e.g., if the Young modulus is given in MPa, so should be the hardening moduli and the yield stress, and similarly for other dimensional parameters of the model.

Table 6-6 lists all parameters required for the definition of an EP material with damage in Digimat, their denomination as well as their corresponding dimensions.

In addition, if the created material is to be used in a coupled simulation with a finite element code, the user will make sure that the chosen unit system is consistent with the one used in the finite element model.

Table 6-6 Parameter names, symbols, dimensions and SI units

Name	Symbol	Dimensions - M, L, T, θ, I	SI units
Young's moduli	E, E_0	$ML^{-1}T^{-2}$	$Pa = N/m^2$
Poisson's ratio	υ	1	-
Yield Stress	σ_Y	$ML^{-1}T^{-2}$	$Pa = N/m^2$
Hardening modulus	k	$ML^{-1}T^{-2}$	$Pa = N/m^2$
Hardening modulus	R_∞	$ML^{-1}T^{-2}$	$Pa = N/m^2$
Hardening exponent	m	1	-
Critical damage	D_c	1	-
Damage exponent	N	1	-
Damage rate factor	$DRF=1/S_0$	$M^{-1}LT^2$	$Pa^{-1} = m^2/N$
Damage initiation threshold	p_D	1	-

Thermo-elastoplasticity

Stress-strain response of a thermo-elasto-plastic (TEP) material exhibits nonlinear behavior as soon as the stress exceeds a threshold value called yield stress, σ_y. Both elastic and plastic part of the stress-strain response can be temperature dependent. If the material is independent of the strain rate, then this material can be modeled using elasto-plasticity theory. If a dependence on the strain rate on the stress/strain response is observed, models based on thermo-elasto-viscoplasticity should be used, see the respective sections.

Thermo-elastoplasticity Model

The TEP constitutive model available in Digimat is derived from the J_2-plasticity model for which each elastic, plastic and thermal parameters are temperature dependent. This model is based on the von Mises equivalent stress σ_{eq}, defined as:

$$\sigma_{eq} = \sqrt{3J_2(\sigma)} = \left(\frac{1}{2}[(\sigma_{11}-\sigma_{22})^2+(\sigma_{22}-\sigma_{33})^2+(\sigma_{33}-\sigma_{11})^2]+3[\sigma_{12}^2+\sigma_{23}^2+\sigma_{31}^2]\right)^{1/2} \quad (6\text{-}79)$$

In this constitutive model, the response is assumed to be linear elastic as long as the following condition is satisfied:

$$\sigma_{eq} \leq \sigma_Y(T), \quad (6\text{-}80)$$

where $\sigma_y(T)$ denotes the initial yield stress of the material which can be temperature dependent.

The total strain observed by the material is assumed to be the sum of the elastic strain, the plastic strain and the thermal strain such as,

$$\varepsilon = \varepsilon^e + \varepsilon^p + \varepsilon^{th} \quad (6\text{-}81)$$

The Cauchy stress, the total strain, the thermal strain and the plastic strain are then related by

$$\sigma = C(T):(\varepsilon-\varepsilon^p)+\beta(T) \text{ such as } \beta(T) = -C(T):\varepsilon^{th}(T) \quad (6\text{-}82)$$

where C is Hooke's operator which can be temperature dependent through the Young modulus and the Poisson's ratio. The thermal strain is isotropic and is defined as a function of the actual temperature T, the reference temperature T_{ref} and the initial temperature T_{ini} as:

$$\varepsilon^{th}(T) = \{\alpha(T)[T-T_{ref}]-\alpha(T_{ini})[T_{ini}-T_{ref}]\}1 \quad (6\text{-}83)$$

where the coefficient of thermal expansion $\alpha(T)$ can be temperature dependent and the reference temperature being the temperature at which the thermal strain is null.

Once σ_{eq} exceeds the initial yield stress, the response becomes nonlinear and plastic deformation appears. A yield function f (σ ,R,T) can be defined,

$$f(\sigma, R, T) = \sigma_{eq} - \sigma_Y(T) - R(p, T) \leq 0 \quad (6\text{-}84)$$

where $R(p,T)$ is the hardening stress which can be temperature dependent and p the accumulated plastic strain.

If f (σ ,R,T) is lower than zero, the material evolves in the elastic domain. Otherwise it is in the plastic region. In this case, the von Mises norm of the Cauchy stress is given by

$$\sigma_{eq} = \sigma_Y(T) + R(p, T) \quad (6\text{-}85)$$

As the isothermal EP model, the evolution of the plastic strain tensor ε^p is given by the normality rule:

$$\dot{\varepsilon}^p = \dot{p}\frac{\partial f}{\partial \sigma} \quad (6\text{-}86)$$

Isotropic hardening model

Three laws are available for the hardening stress:

- Power law,

$$R(p, T) = k(T)p^{m(T)} \quad (6\text{-}87)$$

- Exponential law,

$$R(p, T) = R_\infty(T)\{1 - \exp[-m(T)p]\} \quad (6\text{-}88)$$

- Exponential and linear law,

$$R(p, T) = k(T)p + R_\infty(T)\{1 - \exp[-m(T)p]\} \quad (6\text{-}89)$$

All parameters of the hardening stress can be temperature dependent.

If the tensile stress/strain curve of the material exhibits a horizontal plateau, the exponential hardening law should be retained. The exponential and linear law is used when a plateau is almost reached but the stress level keeps increasing slowly.

Isotropic extraction method

In the homogenization process of TEP materials, the isotropic part of the tangent stiffness tensor must be extracted. Three different methods are available in Digimat. They are described in detail in Chapter 5, Isotropic Extraction Methods.

Units

Digimat-MF is not bound to a particular system of units. Nevertheless, units must be consistent throughout the definition of the material model, e.g., if the Young modulus is given in MPa, so

should be the hardening moduli and the yield stress, and similarly for other dimensional parameters of the model.

Table 6-7 lists all the parameters introduced in this page, their denomination as well as their corresponding dimensions.

In addition, if the created material is to be used in a coupled simulation with a finite element code, the user will make sure that the chosen unit system is consistent with the one used in the finite element model.

Table 6-7 Parameter names, symbols, dimensions and SI units

Name	Symbol	Dimensions - M, L, T, θ, I	SI units
Young's modulus	E	$ML^{-1}T^{-2}$	$Pa = N/m^2$
Poisson's ratio	ν	1	-
Yield Stress	σ_Y	$ML^{-1}T^{-2}$	$Pa = N/m^2$
Hardening modulus	k	$ML^{-1}T^{-2}$	$Pa = N/m^2$
Hardening modulus	R_∞	$ML^{-1}T^{-2}$	$Pa = N/m^2$
Hardening exponent	m	1	-
Coefficient of Thermal expansion	α	θ^{-1}	K^{-1}

Elasto-viscoplasticity

It is commonly observed that the loading rate affects the mechanical response of some materials, especially in the field of plastics and composites. This is caused by the so-called viscous effects. These can be decomposed more specifically into viscoelastic and viscoplastic effects. A variation of the strain rate does in such case result in a change on the apparent stiffness of the material.

In the same material, the intensity level of the viscoelastic and the viscoplastic effects can be relatively different. In other words it is possible that a material shows a negligible viscoelastic behavior while being sensitive to the strain rate in the plasticity zone. In such case we simply consider the material as being elasto-viscoplastic (EVP), i.e., that the elastic response is not rate-dependent while the plastic one is.

Taking into account the strain-rate dependency in the mechanical behavior law is of great importance when addressing many industrial problems like impact tests, crash tests, high rotational velocity loadings, etc. The objective of this section is to describe how Digimat addresses viscoplastic effects and the models available to describe them. Some guidelines and limitations will also be proposed to help you define your own EVP material law.

Theory

As shown for an EP material (see section Elasto-plasticity: J_2-plasticity Model), the stress-strain curve of a material is divided into two main domains: elastic and plastic. Such division of the stress-strain curve can be done similarly for an EVP material, as you get an elastic domain followed by a viscoplastic one. The threshold that separates these two domains is the yield stress of the material, as for EP materials. For the EVP material, the yield stress does not vary as a function of the strain rate, as it is the case for the fully coupled viscoelastic-viscoplastic (VE-VP) material (see section Viscoelasticity-viscoplasticity).

Let us also recall that the elastic domain is characterized by a totally reversible strain. Whether this reversibility is instantaneous or not depends on whether the material is elastic (see section Linear (thermo-)elasticity or viscoelastic (see section Viscoelasticity) although for the EVP material, we do not account for viscous effects in the elastic domain of deformation. If the applied load exceeds the yield stress, the material then goes into plasticity and the deformation is no longer reversible but said to be permanent. Thus the total strain (ε) is said to be the sum of a viscoelastic part (ε^e) with a viscoplastic one (ε^p)

$$\varepsilon = \varepsilon^e + \varepsilon^p, \ \varepsilon_{ij} = \varepsilon^e_{ij} + \varepsilon^p_{ij} \tag{6-90}$$

In a general way, the viscous effects can be illustrated as a result of the time left to the material to conform itself with a new position, at a molecular level, in order to accommodate a given load, using a minimum of energy. For plastics, it is said that if the load is applied at a very low strain rate (quasi-static test), the molecular chains have time to reorganize and align themselves accordingly to the load. However, if the same load is applied at a higher strain rate, the molecular chains do not necessarily have sufficient time to completely reorganize themselves and, as a general result, a stiffer mechanical response is observed.

For the EVP materials, the viscous effects are only considered in the plastic domain as in the elastic region, the elastic behavior is considered to be independent of the strain rate. Isotropic elasticity is assumed and entirely defined with 2 parameters being the elastic modulus and the Poisson's ratio of the material. For more details on the elastic model, refer to the documentation section treating elastic materials (see section Linear (thermo-)elasticity).

Viscoplasticity

Viscoplasticity treats the domain in which strains are no longer reversible but said to be permanent. To verify that a material exhibits a viscoplastic behavior, one simply has to make a monotonic tensile test under imposed stress beyond the initial yield stress, maintain this stress for a while, and then suddenly unload the specimen until zero stress. If after a very large time there is still some permanent strain in the specimen, then the material exhibits a viscoplastic response.

Similarly to viscoelasticity, the strain rate effect is translated into an apparent stiffening of the material as the strain rate increases. This explains why in impact tests the stress a composite material withstands before failure is generally greater at high strain rates than that for a quasi-static test.

Another way to illustrate the viscoplastic behavior is to apply and maintain a fixed load larger than the yield stress. It can be shown that, in such a case, on a viscoplastic material, the deformation increases slowly but continuously. This is called a creep behavior and it is governed by the viscoplastic properties of the material. Notice this is also observable in the viscoelastic regime.

Before defining any viscoplastic model on a material, it must be pointed out that the plastic behavior of the material should first be modeled, based on curves at very low strain rate. As explained in the documentation page on the EP model (see section Elasto-plasticity: J_2 plasticity Model), the material can show isotropic hardening. For the homogenization of composite materials with a EVP phase, an isotropic extraction method must also be chosen in combination with the hardening model chosen (see Chapter Isotropic Extraction Methods). All of that has to be done as a first approach of the viscoplastic model identification, using quasi-static loadings.

Then, the way viscoplasticity is expressed can differ depending on the material. In order to address a wide range of viscoplastic behaviors, different laws are proposed in Digimat and described below. The viscoplastic laws are written in such a way that they define the evolution of the plastic strain after yielding.

Before going on with the description of the different viscoplastic and creep laws, let us define some variables and some parameters:

- σ_y is the material initial yield stress
- $R(p)$ is the hardening stress (plastic contribution) that is calculated from the hardening law (see section Elasto-plasticity: J_2 plasticity Model)
- f is the viscoplastic part of the stress. It is defined as

$$f = \sigma_{eq} - \sigma_Y - R(p) \tag{6-91}$$

where σ_{eq} is the von Mises equivalent stress.

- dp/dt is the rate of plastic deformation, also called the accumulated plastic strain rate. Notice we will also use the following notation:

$$\frac{dp}{dt} = \dot{p} \tag{6-92}$$

- η is the viscoplastic coefficient of the Norton, hyperbolic sinus and power laws. This parameter expresses the viscoplastic sensitivity of the material to the strain rate.
- m is the viscoplastic exponent for all four viscoplastic laws.

The evolution of the viscoplastic strain tensor obeys the following flow rule:

$$\dot{\varepsilon} = \dot{p}\frac{\partial f}{\partial \sigma} \tag{6-93}$$

Notice that the following viscoplastic laws are valid only when the following conditions are met:

$$\dot{p} = 0 \quad \text{if} \quad f \leq 0,$$

$$\dot{p} > 0 \quad \text{if} \quad f > 0 \tag{6-94}$$

In Digimat, several viscoplastic laws are available but a distinction is made depending on the type of experiments:

- High strain rate sensitivity: Usually used to describe the strain rate effect on the plastic behavior of such materials
  - Initial yield Norton law
  - Current yield Norton law
  - Hyperbolic sinus law (Prandtl law)
- Creep model: Usually used to describe creep behavior of such materials
  - Initial yield Norton law
  - Current yield Norton law
  - Hyperbolic sinus law (Prandtl law)
  - Power law
  - Time law

Initial Yield Norton Law

$$\dot{p} = \frac{\sigma_Y}{\eta}\left(\frac{f}{\sigma_Y}\right)^m \tag{6-95}$$

Current Yield Norton Law

$$\dot{p} = \frac{\sigma_Y}{\eta}\left(\frac{f}{\sigma_Y + R(p)}\right)^m \tag{6-96}$$

At first look, it is true to say that the two Norton laws are very similar. In the initial yield Norton law, the viscoplastic stress (f) depends only on the accumulated plastic strain rate and the initial yield stress σ_y which is constant. However, in the current yield Norton law, f depends on the accumulated plastic strain rate, the yield stress σ_y and the hardening stress $R(p)$. This difference is important because it means that the viscoplastic stress is being updated as the hardening stress increases. This second formulation is more commonly used than the first one and is recommended.

As the strain rate increases, the material stiffens in such a way that the viscoplastic behavior becomes more and more linear for both Norton laws.

Hyperbolic Sinus Law (Prandtl Law)

$$\dot{p} = \frac{\sigma_Y}{\eta}\left(\sinh\left(\frac{f}{\beta}\right)\right)^m \tag{6-97}$$

Here, β is defined as the second viscoplastic coefficient. It has essentially the same kind of effect as the viscoplastic coefficient but to a different degree of sensibility.

As for the initial yield Prandtl law, the viscoplastic stress in the hyperbolic sinus law depends on the yield stress but not on the hardening stress. This law however shows a different type of strain rate dependency from the Norton laws since, instead of heading towards a linear mechanical response at high strain rates, the shape of the curve keeps essentially being the same as that of a quasi-static loading example. That leads to say that for different strain rates, the curves all have similar nonlinear shapes but stack one above another as they reach different maximum levels of stress. It means that the hyperbolic sinus law allows to get saturation in stress even at high strain rates.

Power Law

$$\dot{p} = \frac{\sigma_Y}{\eta}\left(\frac{\sigma_{eq}}{\sigma_Y + R(p)}\right)^m \tag{6-98}$$

This law has a very similar form to the current yield Norton law, the difference lying in the fact that the accumulated plastic strain rate this time depends upon the von Mises stress instead of the viscoplastic stress.

Remark: Up to now, this power law formulation of viscoplasticity has not demonstrated very good correlations on anisotropic viscoplastic materials using the homogenization approach of Digimat. It is also more prone to convergence issues. Therefore, this viscoplastic law is not recommended for common use.

Time Law

$$\dot{p} = A\,\sigma_{eq}^{m}\;t^{n} \tag{6-99}$$

The time law is a creep law which depends on time and on the von Mises stress. Three parameters are needed to completely define this creep law. The range of values for the three parameters A, n and m respectively called the creep coefficient, the creep exponent and the second creep exponent are: $A > 0$, $m > 0$ and $-1 \quad n < 0$. Contrary to the previously defined viscoplastic laws, the time law does not depend on the yield stress and hardening parameters.

Units

Digimat-MF is not bound to a particular system of units. Nevertheless, units must be consistent throughout the definition of the material model, e.g., if the Young modulus is given in MPa, so should be the hardening moduli and the yield stress, and similarly for other dimensional parameters of the model.

Table 6-8 lists all the parameters introduced in this page, their denomination as well as their corresponding dimensions.

Table 6-8 Parameter names, symbols, dimensions and SI units

Name	Symbol	Dimensions - M, L, T, θ, I	SI units
Density	ρ	ML^{-3}	kg/m^3
Young's modulus	E	$ML^{-1}T^{-2}$	$Pa = N/m^2$
Poisson's ratio	ν	1	-
Yield Stress	σ_Y	$ML^{-1}T^{-2}$	$Pa = N/m^2$
Hardening modulus	R_∞	$ML^{-1}T^{-2}$	$Pa = N/m^2$
Hardening exponent	n	1	-
Hardening modulus 2	k	$ML^{-1}T^{-2}$	$Pa = N/m^2$
Creep coefficient	η	$ML^{-1}T^{-2}$	$Pa.s = N/m^2.s$
Creep exponent	m	1	-
Creep coefficient 2	β	$ML^{-1}T^{-2}$	$Pa = N/m^2$

Crystal Plasticity

The crystal plasticity model is extensively described in section Crystal Plasticity.

Thermo-elasto-viscoplasticity

It is commonly observed that the mechanical and thermal loading rate affects the mechanical response of some materials, especially in the field of plastics and composites.

This section proposes to describe a thermo-elasto-viscoplastic material model which exhibits viscous effect in the (thermo)plastic part and which is strain rate insensitive in the thermo-elastic domain. For this model, all parameters can be temperature dependent: elastic, thermal, plastic and viscoplastic.

Taking into account the rate dependency in the mechanical behavior law and the temperature effect is of great importance when addressing many industrial problems like impact tests, crash tests, high rotational velocity loadings, creep test, etc... The objective of this section is to present this material model of a theoretical and practical point view. define your own thermo-elasto-viscoplastic material law.

Theory

As shown for a thermo-elastoplastic material (see section Thermo-elastoplasticity), the total strain is divided into three main domains: elastic (ε^e), thermal (ε^{th}) and plastic (ε^p):

$$\varepsilon = \varepsilon^e + \varepsilon^p + \varepsilon^{th} \tag{6-100}$$

Such division of the total strain can be done similarly for a thermo-elastoviscoplastic material. The stressstrain curve of such material is also divided in two main domains: a thermo-elastic domain followed by a (thermo)viscoplastic one. The threshold that separates these two domains is the yield stress of the material σ_y (T), as for thermo-elastoplastic materials, which can also depend on the temperature field and not of the strain rate, as it is the case of the fully coupled viscoelastic-viscoplastic material (see section Viscoelasticity-viscoplasticity).

The Cauchy stress, the total strain, the thermal strain and the plastic strain are then related by

$$\sigma = C(T) : (\varepsilon - \varepsilon^p) + \beta(T) \text{ such as } \beta(T) = -C(T) : \varepsilon^{th}(T) \tag{6-101}$$

where C(T) is Hooke's operator which can be temperature dependent through the Young modulus and the Poisson's ratio. The thermal strain is isotropic and is defined as a function of the actual temperature T, the reference temperature T_{ref} and the initial temperature T_{ini} as:

$$\varepsilon^{th}(T) = \{\alpha(T)[T - T_{ref}] - \alpha(T_{ini})[T_{ini} - T_{ref}]\}1 \tag{6-102}$$

where the coefficient of thermal expansion α (T) can be temperature dependent and the reference temperature being the temperature at which the thermal strain is null.

The thermo-elastic domain (i.e., $\varepsilon^p = 0$) is characterized by a totally reversible strain, i.e., we do not account for viscous effects in the elastic domain of deformation. In this domain, isotropic thermo-elasticity is assumed and entirely defined with three parameters being the elastic modulus, the Poisson's ratio and the thermal expansion coefficient of the material (see thermo-elastic section for more details). If the applied load exceeds the yield stress, the material then goes into plasticity and the deformation is no longer reversible but said to be permanent. When plasticity occurs and its evolution is governed by the plastic flow rule:

$$\dot{\varepsilon}^p = \dot{p}\frac{\partial f}{\partial \sigma} \tag{6-103}$$

The precise sign of the rate of plastic deformation dp/dt is determined by the following conditions:

$$\dot{p} = 0 \quad \text{if} \quad f \le 0,$$

$$\dot{p} = g_v(f) > 0 \quad \text{if} \quad f > 0 \tag{6-104}$$

where $g_v(f)$ denotes the viscoplastic law. Some analytical law will be described latter. f(σ, p, T) is the yield function defined as:

$$f(\sigma, R, T) = \sigma_{eq} - \sigma_Y(T) - R(p, T) \leq 0 \quad (6\text{-}105)$$

R(p, T) is the hardening stress (plastic contribution) that is calculated from the hardening law (see section Isotropic Hardening Model). All parameters of the hardening stress can be temperature dependent. Please refer to the corresponding section of the documentation for more details, where σ_{eq} refers to the von Mises equivalent stress (see section Elasto-plasticity: J_2-plasticity Model).

Viscoplasticity and Creep Models

Viscoplasticity treats the domain in which strains are no longer reversible but said to be permanent. For thermo-elasto-viscoplastic material model, viscoplastic domain can be temperature dependent (see section Elasto-viscoplasticity).

In Digimat, several viscoplastic laws are available but a distinction is made depending on the type of experiments:

- High strain rate sensitivity: Usually used to describe the strain rate effect on the plastic behavoir of such materials
  - Initial yield Norton law
  - Current yield Norton law
  - Hyperbolic sinus law (Prandtl law)
- Creep model: Usually used to describe creep behavior of such materials
  - Initial yield Norton law
  - Current yield Norton law
  - Hyperbolic sinus law (Prandtl law)
  - Power law
  - Time law

Initial Yield Norton Law

$$\dot{p} = \frac{\sigma_Y(T)}{\eta(T)}\left(\frac{f}{\sigma_Y(T)}\right)^{m(T)} \quad (6\text{-}106)$$

Current Yield Norton Law

$$\dot{p} = \frac{\sigma_Y(T)}{\eta(T)}\left(\frac{f}{\sigma_Y(T) + R(p, T)}\right)^{m(T)} \quad (6\text{-}107)$$

Hyperbolic Sinus Law (Prandtl Law)

$$\dot{p} = \frac{\sigma_Y(T)}{\eta(T)}\left[\sinh\left(\frac{f}{\beta(T)}\right)\right]^{m(T)} \quad (6\text{-}108)$$

Here, $\beta(T)$ is defined as the second viscoplastic coefficient. It has essentially the same kind of effect as the viscoplastic coefficient but to a different degree of sensibility. Effect of this model coefficients can be found in section Hyperbolic Sinus Law (Prandtl Law).

Power Law

$$\dot{p} = \frac{\sigma_Y(T)}{\eta(T)}\left(\frac{\sigma_{eq}}{\sigma_Y(T) + R(p, T)}\right)^{m(T)} \quad (6\text{-}109)$$

Remark: Up to now, this power law formulation of viscoplasticity has not demonstrated very good correlations on anisotropic viscoplastic materials using the homogenization approach of Digimat. It is also more prone to convergence issues. Therefore, this viscoplastic law is not recommended for common use.

Time Law

$$\dot{p} = A(T)\sigma_{eq}^{m(T)} t^{n(T)} \quad (6\text{-}110)$$

The time law is a creep law which depends on time and on the von Mises stress. Three parameters are needed to completely define this creep law. The range of values for the three parameters A, m and n respectively called the creep coefficient, the creep exponent and the second creep exponent are: A $>$ 0, m $>$ 0 and -1 n $<$ 0. Contrary to the previously defined viscoplastic laws, the time law does not depend on the yield stress and hardening parameters.

Units

Digimat-MF is not bound to a particular system of units. Nevertheless, units must be consistent throughout the definition of the material model, e.g., if the Young modulus is given in MPa, so should be the hardening moduli and the yield stress, and similarly for other dimensional parameters of the model.

Table 6-9 lists all the parameters introduced in this page, their denomination as well as their corresponding dimensions.

Table 6-9 Parameter names, symbols, dimensions and SI units

Name	Symbol	Dimensions - M, L, T, θ, I	SI units
Density	ρ	ML^{-3}	kg/m^3
Young's modulus	E	$ML^{-1}T^{-2}$	$Pa = N/m^2$
Poisson's ratio	ν	1	-
Yield Stress	σ_Y	$ML^{-1}T^{-2}$	$Pa = N/m^2$
Hardening modulus	R_∞	$ML^{-1}T^{-2}$	$Pa = N/m^2$

Table 6-9 Parameter names, symbols, dimensions and SI units

Name	Symbol	Dimensions - M, L, T, θ, I	SI units
Hardening exponent	n	1	-
Hardening modulus 2	k	$ML^{-1}T^{-2}$	$Pa = N/m^2$
Creep coefficient	η	$ML^{-1}T^{-2}$	$Pa.s = N/m^2.s$
Creep exponent	m	1	-
Creep coefficient 2	β	$ML^{-1}T^{-2}$	$Pa = N/m^2$
Coefficient of Thermal expansion	α	θ^{-1}	K^{-1}

Viscoelasticity

It is commonly observed that the loading rate affects the mechanical response of some materials, especially in the field of plastics and composites. This is caused by the so-called viscous effects. These can be decomposed more specifically into viscoelastic and viscoplastic effects. A variation of the strain rate does in such case result in a change on the apparent stiffness of the material.

The viscoelastic (VE) model is of great importance to capture the dependence of a material's elastic response with respect to the loading rate. Such a material model is often necessary to treat vibration tests, cyclic loading and so on. The objective of this documentation section is to describe how Digimat treats the viscoelastic behavior. Some guidelines and limitations will also be proposed to help the user define his/her own viscoelastic material law.

Elastic strain must be seen as an instantaneous strain. However, when talking about viscoelasticity it means that for a given load, part of the strain is instantaneous and part of it is delayed in time. This second part belongs to the viscous behavior. Another way of seeing such a behavior is to make a monotonic tensile test under imposed stress, maintain this stress for a while, and then suddenly unload the specimen until zero stress.

Due to viscoelasticity, a full recovery of the initial strain is not instantaneous but takes a certain time.

In a general way, viscous effects can be illustrated as a result of the time left to the material to conform itself with a new position, at a molecular level, in order to accommodate a given load, using a minimum of energy. For plastics, it is said that, if the load is applied at a very low strain rate (quasi-static test), the molecular chains have time to reorganize and align themselves according to the load. However, if the same load is applied at a higher strain rate, the molecular chains do not necessarily have sufficient time to completely reorganize themselves and, as a general result, a stiffer mechanical response is observed.

At the elastic level, the elasticity can be partly linear and partly nonlinear, depending on the strain rate and the relaxation times proper to the material. The higher the loading strain rate and the higher the stiffness gets because the material has less time to relax. Notice that Digimat models only the linear elastic strain rate dependency.

Theory

Mathematically the linear viscoelastic constitutive model is defined as

$$\sigma(t) = G(t) : \varepsilon(0) + \int G(t-r) : \dot{\varepsilon}^{(ve)}(\tau)dr \tag{6-111}$$

with $\varepsilon(0) = \lim_{\substack{t \to 0 \\ t > 0}} \varepsilon(t)$ and $G(t) = 2G_R(t)I^{dev} + K_R(t)1 \otimes 1$

The integral equation represents the memory effect. This means that the stress σ_t(t) at time > 0 depends on all the strain history up to that time, i.e., σ_t(t) depends on ε(s), $s \leq t$.

G(t) are the relaxation moduli, which in the isotropic case are represented by time-dependent shear and bulk moduli $G_R(t)$ and $K_R(t)$, respectively. That leads to introduce isotropic viscoelasticity in Digimat as Prony series of the shear modulus G(t) and of the bulk modulus *K(t)*, which are defined in the following way:

$$G_{R(t)} = G_0\left[1 - \sum_{i=1}^{n} w_i\left(1 - e^{-t/\tau_i}\right)\right],\ G_0 = G(t=0) \tag{6-112}$$

$$K_{R(t)} = K_0\left[1 - \sum_{i=1}^{n'} w_i^*\left(1 - e^{-t/\tau_i^*}\right)\right],\ K_0 = K(t=0) \tag{6-113}$$

G_0 is defined as the Initial shear modulus of the viscoelastic material. This parameter expresses the shear modulus of the material at t = 0 of a relaxation test. It can also be seen as the shear modulus of the material if it was loaded at an infinite strain rate. It has the dimension of a stress for which the units simply have to be in accordance with the global units used in the overall material model.

K_0 is defined as the Initial bulk modulus of the viscoelastic material. This parameter expresses the compressibility modulus of the material at t = 0 of a relaxation test. It can also be seen as the bulk modulus of the material if it was loaded at an infinite strain rate. It has the dimension of a stress for which the units simply have to be in accordance with the global units used in the overall material model.

To ensure a correct understanding of the bulk modulus, it must be understood as the resistance of a material to a uniform pressure. It is defined as the pressure increase required to cause a given relative decrease in volume.

τ_i and τ_i^* are defined as the relaxation times of the material; w_i and w_i^* are defined as the weight of each given relaxation time.

Table 6-10 Range of available values for Prony series parameters

Relevant parameters	Prony series for shear modulus G	Prony series for bulk modulus K
Shear / Bulk modulus	$G_0 > 0$	$K_0 > 0$
Shear / Bulk relaxation time	$\tau_i > 0$	$\tau_i^* > 0$
Shear / Bulk weight	$1 > w_i \geq 0$	$1 > w_i^* \geq 0$

Remark: It is important to mention that the shear modulus G(t) and the bulk modulus K(t) must be strictly positive, i.e.,

$$\sum_{i=1}^{n} w_i < 1 \text{ and } \sum_{i=1}^{n'} w_i^* < 1 \tag{6-114}$$

Each weight wi is a measure of the sensitivity of the material to a given strain rate. If the weight of a given relaxation time is very low, that means the strain rate dependency around this relaxation time is limited, i.e., the variation of the stiffness is limited. As the weight value increases, the sensitivity of the material to the strain rate should continuously increase.

Note that for a very large (infinite) time, the time-dependent shear and bulk moduli tend towards the classical linear elastic ones.

$$G_\infty(t) = G_0\left(1 - \sum_{i=1}^{n} w_i\right),\ K_\infty(t) = K_0\left(1 - \sum_{i=1}^{n'} w_i^*\right) \tag{6-115}$$

In Digimat, the definition of a viscoelastic material should consist of

- a density
- a shear and a bulk moduli for a relaxation time that tends towards 0 (instantaneous moduli)
- a table defining the Prony series of the shear modulus (i.e., pairs of a shear weight corresponding to a relaxation time)
- a table defining the Prony series of the bulk modulus (i.e., pairs of a bulk weight corresponding to a relaxation time).

Definition from Storage and Loss Moduli

A viscoelastic material can also be defined from storage and/or loss moduli functions. A function fitting is then performed to define the instantaneous moduli and Prony series tables of the shear and bulk moduli.

To enable this option, the Storage and loss moduli input method must be selected from the material Model tab after choosing a Viscoelastic material model. Two additional areas appear in the material Parameters tab as compared to the direct viscoelastic parameters input method (see Figure 6-8 or Figure 6-9).

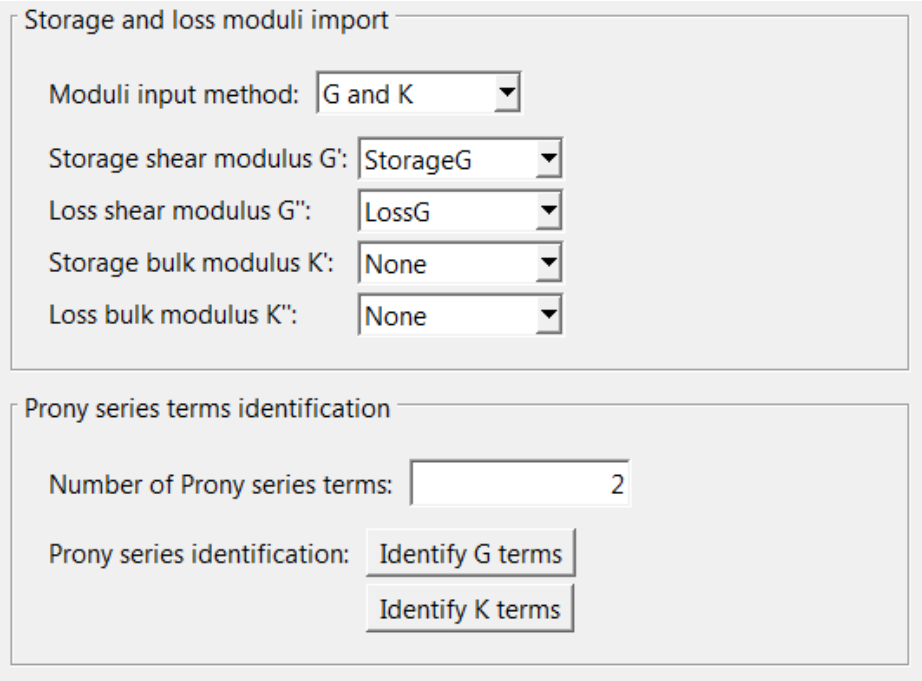


Figure 6-8 Viscoelastic material definition: Shear and Bulk moduli input method.

Storage and loss moduli import. Two moduli input methods are available: either **G and K** or **E and Nu** methods. In the **G and K** method (see Figure 6-8), the shear and bulk instantaneous moduli and Prony series terms are identified separately. The storage and loss modulus functions must be selected for each of the moduli. For the **E and Nu** method (see Figure 6-9), the shear and bulk instantaneous moduli and Prony series terms are identified simultaneously from the selected storage and loss Young modulus functions, and the Poisson's ratio. Moduli functions must be defined in the function manager (**Tools** > **Manage Functions**) to be available in the selection list. The storage functions selection is mandatory whereas loss functions are optional.

Prony series terms identification. This area control the number of Prony series terms requested for calibration and the identification procedure launch. The minimum Prony series terms number that may be entered is 2, and the maximum is 10. In the **G and K** input method, the **Identify G terms or Identify K terms** button must be hit to execute the fitting procedure for the shear and bulk functions, respectively. The **Identify terms** button must be hit in the **E and Nu** input method case.

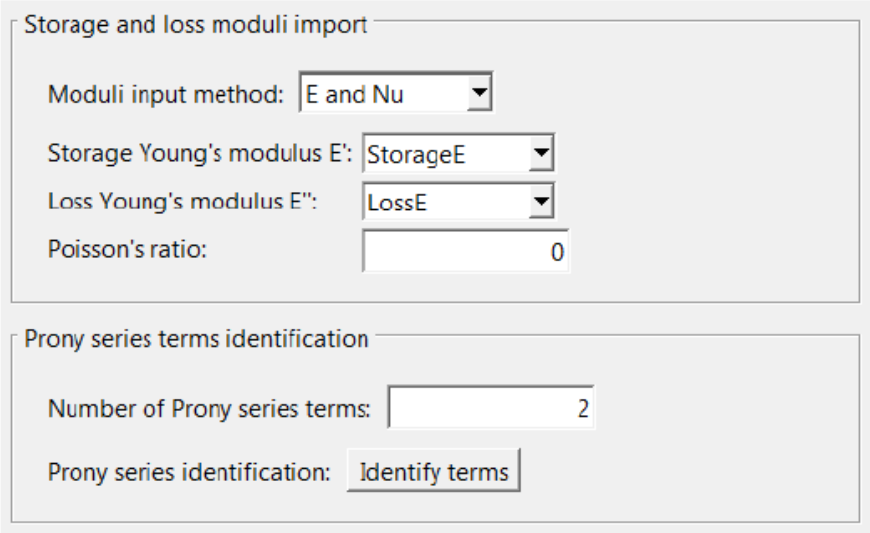


Figure 6-9 Viscoelastic material definition: Young modulus and Poisson's ratio input method.

The fitting procedure consists on nested global and local optimizations and may last up to one minute depending on the function definition and the requested Prony series terms number. A progress bar notify the user of the procedure advancement.

As Prony series terms fitting is a complex procedure, the optimization may lead to a local optimum which is not the global optimum. The optimization parameters are tuned such that the global optimum will be reach in most of the case. However, for specific modulus functions, the global optimum may be difficult to reach. In that case, several calibration runs may be needed to reach the global optimum. To ease this procedure where calibrations are run successively on the same data, newly obtained Prony parameters will only be kept if they lead to a better solution than previously obtained parameters.

Units

Digimat-MF is not bound to a particular system of units. Nevertheless, units must be consistent throughout the definition of the material model, e.g., if the Young modulus is given in MPa, so should be the hardening moduli and the yield stress, and similarly for other dimensional parameters of the model.

Table 6-11 lists all the parameters introduced in this page, their denomination as well as their corresponding dimensions.

In addition, if the created material is to be used in a coupled simulation with a finite element code, the user will make sure that the chosen unit system is consistent with the one used in the finite element model.

Table 6-11 Parameter names, symbols, dimensions and SI units

Name	Symbol	Dimensions - M, L, T, θ, I	SI units
Density	ρ	ML^{-3}	kg/m^3
Shear modulus	G	$ML^{-1}T^{-2}$	$Pa = N/m^2$
Bulk modulus	K	$ML^{-1}T^{-2}$	$Pa = N/m^2$
Prony weight	w_i	1	-
Relaxation time	τ_i	T	s

Thermo-viscoelasticity

The thermo-viscoelastic (TVE) model is of great importance to capture the dependence of a material elastic response with respect to the loading rate and the thermal loading. Such a material model is often necessary to treat vibration tests, cyclic loading, curing and so on. The objective of this documentation section is to describe how Digimat treats the TVE behavior and the different shift functions available in Digimat to account for thermal effects. to help the user define his/her own viscoelastic material law.

Theory

The TVE model is based on the assumption that the total strain ε is split into a viscoelastic part ε^{ve} and a thermal part ε^{th} as:

$$\varepsilon = \varepsilon^{ve} + \varepsilon^{th} \tag{6-116}$$

The following definition is used for the thermal strain ε^{th}:

$$\varepsilon^{th} = [\alpha(T)(T - T_{ref}) - \alpha(T_{ini})(T_{ini} - T_{ref})]1 \tag{6-117}$$

where T denotes the temperature field, T_{ref} the reference temperature and T_{ini} the initial temperature. 1 denotes the second order identity tensor. The reference temperature being the temperature at which the thermal strain is null. α(T) denotes the thermal expansion coefficient and is temperature dependent. Only an isotropic definition of the thermal strain is available in Digimat for such materials. The Cauchy stress is related to the history of the viscoelastic strain through the Boltzmann's hereditary integral (following the linear viscoelasticity theory):

$$\sigma(t) = \int_{-\infty}^{t} E(\tau - \tau', T) : \frac{\partial \varepsilon^{ve}}{\partial t'} dt' \tag{6-118}$$

where τ denotes the reduced time and is discussed in details below. The relaxation tensor E(t, T) is assumed to be isotropic:

$$E(t, T) = 3K_R(t, T)J + 2G_R(t, T)K \tag{6-119}$$

E(t, T) are the relaxation moduli which, in the isotropic case, are represented by time-dependent shear and bulk moduli G_R(t, T) and K_R(t, T), respectively. J and K are two orthogonal projection tensor which project any symmetric second order tensor on its spherical and deviatoric parts respectively:

$$\begin{cases} J_{ijkl} = \frac{1}{3}1_{ij}1_{kl} \\ I_{ijkl} = J_{ijkl} + K_{ijkl} \\ J_{ijkl} : \mathrm{K}_{ijkl} = 0 \end{cases} \tag{6-120}$$

That leads to introduce isotropic viscoelasticity in Digimat as Prony series of the shear modulus G_R(t, T) and of the bulk modulus K_R(t, T), which are defined in the following way:

$$G_R(t, T) = G_0(T)\left[1 - \sum_{i=1}^{n} w_i(1 - \exp(-t/\tau_i))\right],\ G_0(T) = G(t = 0, T) \tag{6-121}$$

$$K_R(t, T) = K_0(T)\left[1 - \sum_{i=1}^{n'} w_i^*(1 - \exp(-t/\tau_i^*))\right],\ K_0(T) = K(t = 0, T) \tag{6-122}$$

G_0 is defined as the initial shear modulus of the viscoelastic material. This parameter expresses the shear modulus of the material at t = 0 of a relaxation test. It can also be seen as the shear modulus of the material if it was loaded at an infinite strain rate. It has the dimension of a stress for which the units simply have to be in accordance with the global units used in the overall material model.

K_0 is defined as the Initial bulk modulus of the viscoelastic material. This parameter expresses the compressibility modulus of the material at t = 0 of a relaxation test. It can also be seen as the bulk modulus of the material if it was loaded at an infinite strain rate. It has the dimension of a stress for which the units simply have to be in accordance with the global units used in the overall material model.

To ensure a correct understanding of the bulk modulus, it must be understood as the resistance of a material to a uniform pressure. It is defined as the pressure increase required to cause a given relative decrease in volume. τ_i and τ_i^* are defined as the relaxation times of the material; w_i and w_i^* are defined as the weight of each given relaxation time.

In the Cauchy stress definition, τ(t) denotes the reduced time and is related to the actual time through the integral differential equation:

$$\tau(t) = \int_0^t \frac{dt'}{A_T(T(t'))} \Rightarrow \frac{d\tau}{dt} = \frac{1}{A_T(T(t))} \tag{6-123}$$

where AT denotes the shift function. To compute this variable we assume a linear variation of h(T) such that:

$$-\ln A_T(T(t)) = h(T) = a + bt \tag{6-124}$$

Then from the differential expression of the reduced time defined over a time increment [t_n, t_{n+1}], it follows:

$$\Delta\tau = \int_{t_n}^{t_{n+1}} \exp(a + bt)dt \tag{6-125}$$

where a and b are solution of the following set of equations:

$$\begin{cases} h(T_n) = a + bt_n \\ h(T_{n+1}) = a + bt_{n+1} \end{cases} \tag{6-126}$$

Finally, after a straightforward calculation, it follows the analytical expression of $\Delta\Gamma$

$$\Delta\Gamma = \frac{A_T^{-1}(T_{n+1}) - A_T^{-1}(T_n)}{h(T_{n+1}) - h(T_n)}\Delta t \tag{6-127}$$

Four different definitions of the shift function A_T (T) are available in Digimat:

- **Williams-Landell-Ferry (WLF)**

  The Williams-Landell-Ferry (WLF) shift function is defined as follows:

$$-\log A_T(T) = -\frac{\ln A_T(T)}{\ln 10} = h(T) = \frac{C_1^g (T - T_g)}{C_2^g + (T - T_g)} \tag{6-128}$$

where T_g denotes the reference temperature and, Cg_1 and Cg_2 are constants. Cg_1, Cg_2 and T_g are constant parameters. A singularity occurs for this particular value of the temperature field:

$$T = T_g - C_2^g \tag{6-129}$$

If $T = T_g - C_2^g$ the viscosity is assumed to be infinite, i.e, the material behavior becomes purely elastic.

Note that the parameters of the Prony series are obtained at the temperature Tg.

- **User Defined**

Another possibility is to allow the user to define his/her own function $\log(A_T)$ as a function of the temperature T based on experimental data.

- **Arrhenius**

  Arrhenius shift function is defined as follows:

$$\ln A_T(T) = \frac{\Delta U}{R}\left(\frac{1}{T - T_z} - \frac{1}{T_0 - T_z}\right) \tag{6-130}$$

where ΔU is the energy activation, R the universal gas constant, T_z the absolute zero in the temperature scale being used (i.e. 0 Kelvin if the model is in Kelvin, -273.15C if the model is in Celsius) and T_0, the reference temperature. By default T_z is equal to 0^0 C and R is equal to 8.314472 $J.K^{-1}.mol^{-1}$.

- **Curing**

  The curing shift function is the Arrhenius shift function which contains an additional term function of the cure degree, X:

$$\ln A_T(T) = \frac{\Delta U}{R}\left(\left(\frac{1}{T - T_z} - \frac{1}{T_0 - T_z}\right) - \left(\frac{1}{T_g(X) - T_z} - \frac{1}{T_g(X_{ref}) - T_z}\right)\right) \tag{6-131}$$

where ΔU is the energy activation, R the universal gas constant, T_z the absolute zero (0 K) in the temperature scale being used, T_0, the reference temperature, X_{ref}, the reference degree of cure and T_g, the glass transition temperature. By default T_z is equal to -273.15^0 C and R is equal to 8.314472 $J.K^{-1}.mol^{-1}$. The other ingredients of the curing model are given in Curing Model.

Table 6-12 Range of available values for Prony series parameters

Relevant parameters	Prony series for shear modulus G	Prony series for shear modulus K
Shear / Bulk modulus	$G_0 > 0$	$K_0 > 0$
Shear / Bulk relaxation time	$\tau_i > 0$	$\tau_i^* > 0$
Shear / Bulk weight	$1 > w_i \geq 0$	$1 > w_i^* \geq 0$

Remark: It is important to mention that the shear modulus *GR(t, T)* and the bulk modulus *KR(t, T)* must be strictly positive, i.e.,

$$\sum_{i=1}^{n} w_i < 1 \quad \text{and} \quad \sum_{i=1}^{n'} w_i^* < 1 \tag{6-132}$$

Each weight w_i is a measure of the sensitivity of the material to a given strain rate. If the weight of a given relaxation time is very low, that means the strain rate dependency around this

relaxation time is limited, i.e., the variation of the stiffness is limited. As the weight value increases, the sensitivity of the material to the strain rate should continuously increase. Note that for a very large (infinite) time, the time-dependent shear and bulk moduli tend towards the classical linear elastic ones.

$$G_\infty(t, T) = G_0(T)\left(1 - \sum_{i=1}^{n} w_i\right) \text{ and } K_\infty(t, T) = K_0(T)\left(1 - \sum_{i=1}^{n'} w_i^*\right) \quad \text{(6-133)}$$

In Digimat, the definition of a viscoelastic material should consist of

- a density
- a shear and a bulk moduli for a relaxation time that tends towards 0 (instantaneous moduli)
- a table defining the Prony series of the shear modulus (i.e., pairs of a shear weight corresponding to a relaxation time)
- a table defining the Prony series of the bulk modulus (i.e., pairs of a bulk weight corresponding to a relaxation time).

When crystallinity model is used, the same Prony Series described in (6-122) are used. In this case, the shift function has a contribution in temperature and in crystallinity content:

$$\tau(t) = \frac{1}{A_T[T(t)] \cdot A_C[T(t)]} \text{ and} \quad \text{(6-134)}$$

$$\log\{A_T[T(t)] \cdot A_C[T(t)]\} = \log\{A_T[T(t)]\} + \log\{A_C[T(t)]\}$$

Where:

- $\log\{A_T[T(t)]\}$ is the Williams-Landell-Ferry (WLF) expression (6-128), or the Arrhenius expression or can be user-defined.
- $\log\{A_C[T(t)]\}$ is the shift function in crystallinity with a WLF expression: $\log(a_c) = \frac{C_1^c(X_c - X_C^{ref})}{C_2^c + (X_c - X_C^{ref})}$. The coefficients C_1^c and C_2^c are parameters calibrated from experimental tests, X_c is the current degree of crystallinity and X_C^{ref} is the reference degree of crystallinity.

Curing Model

The curing model is described by three different laws: the kinetic reaction, the glass transition temperature evolution and the cure shrinkage law.

- The kinetic reaction law is the equation which relates the cure rate dX/dt to the temperature and degree of cure X. The degree of cure is then computed by

$$X(t) = X_0 + \int_0^t \frac{dX}{dt}(X, T)dt \tag{6-135}$$

Three different kinetic reaction laws are available in Digimat:

The Johnston-Hubert (1997) law:

$$\frac{dX}{dt}(X, T) = \frac{KX^m(1-X)^n}{1 + \exp[C(X - X_c)]} \tag{6-136}$$

where

$$K = A\exp\left(-\frac{\Delta E}{R(T - T_2)}\right) \tag{6-137}$$

C is the diffusion constant and

$$X_c = X_{c0} + X_{cT}T \tag{6-138}$$

A is a coefficient, ΔE is the curing activation energy, m and n are exponents, X_{c0} is the initial critical cure degree and X_{ct} the critical cure degree increase. The different parameters can be fitted from the cure degree evolution for various cure cycles. R is the universal gas constant, and T_z is the value of the absolute zero temperature in the current unit system (defined in the shift function section, see Equation (6-131)).

- The model developed by Lee, Chiu and Lin (Lee et al., (1992)):

$$\frac{dX}{dt}(X, T) = K_1(1-X)^n + K_2X^m(1-X)^p \tag{6-139}$$

$$K_1 = A_1 exp\left(-\frac{\Delta E_1}{R(T - T_z)}\right) \tag{6-140}$$

$$K_2 = A_2 exp\left(-\frac{\Delta E_2}{R(T - T_z)}\right) \tag{6-141}$$

A_1 and A_2 are coefficients, ΔE_1 and ΔE_2 are curing activation energies, m, n and p are exponents. The different parameters can be fitted from the cure degree evolution for various cure cycles. R is the universal gas constant, and Tz is the value of the absolute zero temperature in the current unit system (defined in the shift function section, see Equation (6-131)). The usage of T_z in the equation implies that the model parameters (*A1*, ΔE_1, etc.) are the same whatever the temperature unit system.

- A generalized version of the model developed by Kamal and Sourour (1973):

$$\frac{dX}{dt}(X, T) = (K_1 + K_2X^{m(T)})(B(T) - X)^{n(T)} \tag{6-142}$$

$$K_1 = A_1 exp\left(-\frac{\Delta E_1}{R(T-T_z)}\right) \quad (6\text{-}143)$$

$$K_2 = A_2 exp\left(-\frac{\Delta E_2}{R(T-T_z)}\right) \quad (6\text{-}144)$$

$$B(T) = B_0 + B_1(T-T_z) + B_2(T-T_z)^2 \quad (6\text{-}145)$$

$$n(T) = n_0 + n_1(T-T_z) + n_2(T-T_z)^2 \quad (6\text{-}146)$$

$$m(T) = m_0 + m_1(T-T_z) + m_2(T-T_z)^2 \quad (6\text{-}147)$$

To ensure numerical stability, the following conditions are enforced during the computation:

$X \leq B \leq 1; n > 0; m > 0$

B_0, B_1, B_2, m_0, m_1, m_2, n_0, n_1 and n_2 are fitting parameters for the quadratic dependence of B, m and n, respectively, with respect to temperature. The usage of max and min functions ensures that these quadratic equations do not lead to unphysical values (negative exponents in particular).

Still, it is recommended to fit the parameters in a temperature range comparable to that of the simulation. A_1 and A_2 are coefficients. ΔE_1 and ΔE_2 are curing activation energies, m, n and p are exponents. The different parameters can be fitted from the cure degree evolution for various cure cycles. R is the universal gas constant, and T_z is the value of the absolute zero temperature in the current unit system (defined in the shift function section, see Equation (6-131)). The usage of T_z in the equation implies that the model parameters (A_1, ΔE_1, B_0, etc.) are the same whatever the temperature unit system.

- The glass temperature transition law gives the evolution of the glass temperature transition in function of the cure degree. Two models are available, either a quadratic evolution:

$$T_g(X) = a_0 + a_0 X + a_0 X^2 \quad (6\text{-}148)$$

where all a_i coefficients are fitting parameters (note that a_0 is the glass transition temperature of the uncured material), or the DiBenedetto model:

$$T_g(X) = T_{g0} + \frac{\lambda X(T_{g\infty} - T_{g0})}{1-(1-\lambda)X}$$

where T_{g0} and $T_{g\infty}$ are the glass transition temperatures of the uncured and cured material, respectively.

- The cure shrinkage law allows to relate the degree of cure to shrinkage strain. The cure shrinkage follows the Johnston law:

$$V_r^S = 0.0 \qquad X < X_{c1} \tag{6-149}$$

$$V_r^S = BX_s + (V_r^{S\infty} - B)X_S^2 \qquad X_{C1} \le X < X_{C2} \tag{6-150}$$

$$V_r^S = V_r^{S\infty} \qquad X \ge X_{C2} \tag{6-151}$$

$$X_s = \frac{X - X_{C1}}{X_{C2} - X_{C1}} \tag{6-152}$$

where $V|_r^{s\infty}$ is the total volumetric resin shrinkage, X_{C1} is the cure degree at shrinkage initiation, X_{C2} is the cure degree at shrinkage stop and B is a linear coefficient.

The usage of the curing capabilities in Digimat-CAE for the prediction of cure-induced distortions entails the following assumptions:

- For thermomechanical analysis, the matrix material must be thermoviscoelastic when hybrid solution is used in the homogenization procedure. Micro solution is available for both thermoviscoelastic matrix and thermoelastic matrix. For coupled thermomechanical analysis, only micro solution can be used when coupling with FEA code.
- Explicit temperature dependence of the material parameters, such as the coefficient of thermal expansion or initial shear modulus, is not supported. The material stiffness depends implicitly on temperature through the shift function. The coefficient of thermal expansion depends on whether the temperature is above or below the glass transition temperature.
- The development of thermal expansion and chemical shrinkage starts when the degree of cure reaches the gelation point X_{gel}. The latter has the same value as the shrinkage initiation X_{C1}.

Units

Digimat-MF is not bound to a particular system of units. Nevertheless, units must be consistent throughout the definition of the material model, e.g., if the shear modulus is given in MPa, so should be the bulk modulus, and similarly for other dimensional parameters of the model.

Table 6-13 lists all the parameters introduced in this page, their denomination as well as their corresponding dimensions.

Table 6-13 Parameter names, symbols, dimensions and SI units

Name	Symbol	Dimensions - M, L, T, θ, I	SI units
Density	ρ	ML^{-3}	kg/m^3
Shear modulus	G	$ML^{-1}T^{-2}$	$Pa = N/m^2$
Bulk modulus	K	$ML^{-1}T^{-2}$	$Pa = N/m^2$
Prony weight	w_i	1	-
Relaxation time	τ_i	T	s

Crystallinity Model

The crystallinity model is described by two different laws: the kinetic reaction and the crystallization shrinkage strain. The model also considers material state related to the phase transition of the matrix.

This model can be used only if the coupled thermal mechanical analysis type is selected.

A crystallization kinetics model may be calibrated to predict the relative evolution of the crystallinity during temperature evolution. The Nakamura model available in Digimat-MF aims at predicting the percentage of the maximum degree of crystallinity reached during the process.

The Nakamura model implemented is described by the following equation:

$$X_i(t, T) = 1 - \exp\left\{-\left[\int_0^t K(\bar{T})d\tau\right]^n\right\} \quad (6\text{-}153)$$

Where, for the Nakamura model parameters:

- $K(\bar{T})$ is the Nakamura constant
- n is the Avrami Index

Two expressions are possible to define the Nakamura constant $K(\bar{T})$:

1. The Hoffman Lauritzen formulation:

$$K(\bar{T}) = (\ln 2)^{\frac{1}{n}}\left[\frac{1}{t_{0.5}(\bar{T})}\right]$$

$$\left[\frac{1}{t_{0.5}(\bar{T})}\right] = K_0 \exp\left[\frac{-U^*}{R(\bar{T}-\bar{T}_\infty)}\right]\exp\left[\frac{-K_g(\bar{T}+\bar{T}_m^0)}{2\bar{T}^2(\bar{T}_m^0-\bar{T})}\right] \text{ with } \bar{T}_x = T_x - T_z$$

With:

- K_g is the nucleation related constant
- T_g is the glass transition temperature
- T_m^0 is the equilibrium melting temperature
- U is the activation energy
- R is the universal gas constant (U and R must be consistent)

Note that, because some parameters are forced to be in a thermodynamic temperature, consistency must be ensured:

- If external T is in Kelvin $\Rightarrow T_z = 0$
- If external T is in degree Celsius $\Rightarrow T_z =$ -273.15
- If external T is in Fahrenheit scale $\Rightarrow T_z =$ -459.67

2. A Weibull formulation:

$$K(T) = \left\{ M_i \frac{k_i}{\lambda_i} \left(\frac{T-\theta}{\lambda_i} \right)^{k_i - 1} \exp\left[-\left(\left(\frac{T-\theta}{\lambda_i} \right)^{k_i} \right) \right] \right. \qquad \text{if T} \geq \theta$$

$$= 0 \qquad \text{if T} < \theta$$

With:

- θ is the translation temperature
- K is the shape parameter
- λ is the scale parameter
- M is the magnitude parameter

The maximum volumetric crystal shrinkage strain must be defined. This is used to compute the volumetric shrinkage strain that takes place in the matrix and follows a linear evolution between 0 and this maximum volumetric shrinkage strain.

Finally, the material state aims at defining all properties related to the phase transitions of the matrix. For each phase transition, melting and crystallization:

- The phase transition temperature (the temperature related to the highest value of the phase transition peak)
- The specific latent heat
- The specific latent heat low temperature (starting temperature of the phase transition peak). It must be lower than the phase transition temperature (not available for the crystallization phase transition when crystallization kinetics are enabled).
- The specific latent heat high temperature (ending temperature of the phase transition peak). It must be larger than the phase transition temperature not available for the crystallization phase transition when crystallization kinetics are enabled).

Viscoelasticity-viscoplasticity

It is commonly observed that the loading rate affects the mechanical response of some materials, especially in the field of plastics and composites. This is caused by the so-called viscous effects. These can be decomposed more specifically into viscoelastic and viscoplastic effects. A variation of the strain rate does in such case result in a change on the apparent stiffness of the material.

Taking into account the strain-rate dependency in the mechanical behavior law is of great importance when addressing many industrial problems like impact tests, crash tests, high rotational velocity loadings and so on. The objective of this documentation section is to describe how Digimat addresses these viscous effects as much as the models that are available to describe them. Some guidelines and limitations will also be proposed to help you define the parameters of the viscoelastic-viscoplastic (VE-VP) material law.

Theory

As shown for an EP material (see section Elasto-plasticity: J_2-plasticity Model), the stress-strain curve of a material is divided into two main domains: elastic and plastic. Such division of the stress-strain curve can be done similarly for a VE-VP material, as you get a viscoelastic domain followed by a viscoplastic one. The threshold that separates these two domains is the yield stress of the material, as for EP materials. However it is more complex to define it on VE-VP materials, and a section in the guidelines is therefore dedicated to explain how to define the yield stress for such materials.

Let's also recall that the elastic domain is characterized by a totally reversible strain. Whether this reversibility is instantaneous or not depends on whether the material is elastic or viscoelastic (see section Viscoelasticity). If the applied load exceeds the yield stress, the material then goes into plasticity and the deformation is no longer reversible but said to be permanent. Thus the total strain ε is said to be the sum of a viscoelastic part (ε^{VE}) with a viscoplastic one (ε^{P})

$$\varepsilon = \varepsilon^{VE} + \varepsilon^{P}, \quad \varepsilon_{ij} = \varepsilon_{ij}^{VE} + \varepsilon_{ij}^{P} \tag{6-154}$$

In a general way, the viscous effects can be illustrated as a result of the time left to the material to conform itself with a new position, at a molecular level, in order to accommodate a given load, using a minimum of energy. For plastics, it is said that if the load is applied at a very low strain rate (quasi-static test), the molecular chains have time to reorganize and align themselves according to the load. However, if the same load is applied at a higher strain rate, the molecular chains do not necessarily have sufficient time to completely reorganize themselves and, as a general result, a stiffer mechanical response is observed.

In order to simplify the explanation of the VE-VP law, it will be addressed in two separate sub-sections. The illustration of each viscous effect on the results will however be integrated into one complete example presented in the example section.

Viscoelasticity

Elastic strain must be seen as an instantaneous strain. However, when talking about viscoelasticity it means that for a given load, part of the strain is instantaneous and part of it is delayed in time. This second part belongs to the viscous behavior. Another way of seeing such behavior is to make a monotonic tensile test under imposed stress, maintain this stress for a while, and then suddenly unload the specimen until zero stress.

Due to viscoelastic effects, a full recovery of the initial strain is not instantaneous but takes some time. If after a very large time the strain does not fully recover, it means there is permanent strain in the specimen and the stress initially applied was beyond the material yield stress.

At the elastic level, the elasticity can be partly linear and partly nonlinear, depending on the strain rate and the relaxation times proper to the material. The higher the loading strain rate, the higher the stiffness gets because the material has less time to relax. Notice that only linear viscoelasticity is implemented in Digimat.

Mathematically the viscoelastic constitutive model is defined as follows:

$$\sigma(t) = G(t) : \varepsilon(0) + \int G(t-T) : \dot{\varepsilon}^{(ve)}(\tau) d\tau \tag{6-155}$$

with $\varepsilon(0) = \lim_{\substack{t \to 0 \\ t>0}} \varepsilon(t)$ and $G(t) = 2G_R(t)\mathrm{I}^{dev} + K_R(t)1 \otimes 1$

As in pure viscoelasticity (VE), the integral equation models a memory effect. The only difference with pure VE is that the integral here represents the history of viscoelastic strains and not the total strains.

Isotropic viscoelasticity is introduced in Digimat as Prony series of the shear modulus $G_R(t)$ and of the bulk modulus $K_R(t)$, which are defined in the following way:

$$G_R(t) = G_0\left[1 - \sum_{i=1}^{n} w_i\left(1 - e^{-t/\Gamma_i}\right)\right],\ G_0 = G(t=0) \tag{6-156}$$

$$K_R(t) = K_0\left[1 - \sum_{i=1}^{n'} w_i^*\left(1 - e^{-t/\Gamma_i^*}\right)\right],\ K_0 = K(t=0) \tag{6-157}$$

G_0 is defined as the **Initial shear modulus** of the viscoelastic material. This parameter expresses the shear modulus of the material at t = 0 of a relaxation test. It can also be seen as the shear modulus of the material if it was loaded at an infinite strain rate. It has the dimension of a stress for which the units simply have to be in accordance with the global units used in the overall material model.

K_0 is defined as the **Initial bulk modulus** of the viscoelastic material. This parameter expresses the compressibility modulus of the material at t = 0 of a relaxation test. It can also be seen as

the bulk modulus of the material if it was loaded at an infinite strain rate. It has the dimension of a stress for which the units simply have to be in accordance with the global units used in the overall material model.

To ensure a correct understanding of the bulk modulus, it must be understood as the resistance of a material to a uniform pressure. It is defined as the pressure increase required to cause a given relative decrease in volume.

Γ_i and Γ_i^* are defined as the relaxation times of the material; wi and w_i^* are defined as the weight of each given relaxation time.

Table 6-14 Range of available values for Prony series parameters

Relevant parameters	Prony series for shear modulus G	Prony series for bulk modulus K
Shear / Bulk modulus	$G_0 > 0$	$K_0 > 0$
Shear / Bulk relaxation time	$\tau_i > 0$	$\tau_i^* > 0$
Shear / Bulk weight	$1 > w_i \geq 0$	$1 > w_i^* \geq 0$

Remark: It is important to mention that the shear modulus G(t) and the bulk modulus K(t) must be strictly positive, i.e.,

$$\sum_{i=1}^{n} w_{i<1} \text{ and } \sum_{i=1}^{n'} w_i^* < 1 \tag{6-158}$$

Each weight w_i is a measure of the sensitivity of the material to a given strain rate. If the weight of a given relaxation time is very low, that means the strain rate dependency around this relaxation time is limited, i.e., the variation of the stiffness is limited. As the weight value increases, the sensitivity of the material to the strain rate should continuously increase.

Note that for a very large (infinite) time, the time-dependent shear and bulk moduli tend towards the classical linear elastic ones.

$$G_\infty(t) = G_0\left(1 - \sum_{i=1}^{n} w_i\right),\ K_\infty(t) = K_0\left(1 - \sum_{i=1}^{n'} w_i^*\right) \tag{6-159}$$

In Digimat, the definition of a viscoelastic material should consist of

- a density;
- a shear and a bulk moduli for a relaxation time that tends towards 0 (instantaneous moduli);
- a table defining the Prony series of the shear modulus (i.e., pairs of a shear weight corresponding to a relaxation time);

- a table defining the Prony series of the bulk modulus (i.e., pairs of a bulk weight corresponding to a relaxation time).

Viscoplasticity

Viscoplasticity treats the domain in which strains are no longer reversible but said to be permanent. To verify that the material enters or not the viscoplastic zone of deformation, one can simply perform a monotonic loading under imposed stress followed by a total unloading step. If a residual deformation is observed, even after a long rest period, a viscoplastic material response should be considered. If, however, no permanent deformations remain after unloading, this means the applied initial stress was below the yield stress and the material was still behaving in its viscoelastic regime.

As for viscoelasticity, the strain rate effect is translated into an apparent stiffening of the material as the strain rate increases. This explains why in impact tests the stress a composite material withstands before failure is generally greater at high strain rates than that for a quasi-static test. Another way to illustrate the viscoplastic behavior is to apply and maintain a fixed load larger than the yield stress. It can be shown that in such a case, on a viscoplastic material, the deformation slowly but continuously increases. This is called a creep behavior and it is governed by the viscoplastic or creep properties of the material.

Before defining any viscoplastic or creep model on a material, it must be pointed out that the plastic behavior of the material should first be modeled, based on curves at very low strain rate. As explained in the documentation page on the EP model (refer Examples and tutorials), the material can show isotropic hardening. For the homogenization of composite materials with a VE-VP phase, an isotropic extraction method (see chapter Elasto-plasticity: J_2-plasticity Model) must also be chosen in combination with the hardening model chosen. All of that has to be done as a first approach of the viscoplastic model identification, using quasi-static loadings.

Then, the way viscoplasticity is expressed can differ depending on the material. In order to address a wide range of viscoplastic behaviors, different laws are proposed in Digimat and described below. The viscoplastic laws are written in such a way that they define the evolution of the plastic strain after yielding.

Before going on with the description of the different viscoplastic and creep laws, let us define some variables and some parameters:

- σ_y is the material initial **yield stress**.
- $R(p)$ is the **hardening stress** (plastic contribution) that is calculated from the hardening low (see section Isotropic Hardening Model).
- f is the viscoplastic part of the stress. It is defined as

  $$f = \sigma_{eq} - \sigma_Y - R_{(p)} \tag{6-160}$$

  where σ_{eq} is the von Mises equivalent stress.
- dp/dt is the **rate of plastic deformation**, also called the **accumulated plastic strain rate**. Notice we will also use the following notation:

$$\frac{dp}{dt} = \dot{p} \tag{6-161}$$

- η is the **viscoplastic coefficient** of the Norton, hyperbolic sinus and power laws. This parameter expresses the viscoplastic sensitivity of the material to the strain rate.
- m is the **viscoplastic exponent** for all four viscoplastic laws.

The evolution of the viscoplastic strain tensor obeys the following flow rule:

$$\dot{\varepsilon}^p = \dot{p}\frac{\partial f}{\partial \sigma} \tag{6-162}$$

Notice that the following viscoplastic and creep laws are valid only when the following conditions are met:

$$\dot{p} = 0 \text{ if } f \leq 0 \tag{6-163}$$

$$\dot{p} > 0 \text{ if } f > 0 \tag{6-164}$$

In Digimat the distinction is made between viscoplastic and creep laws. Thus, four viscoplastic laws (a.k.a High Strain Rate laws), and five creep laws are implemented:

- High Strain Rate Laws (usually used to describe the strain rate effect on the plastic behavior of such materials):
  - Initial yield Norton law
  - Current yield Norton law
  - Hyperbolic sinus law (Prandtl law)
  - Power law
- Creep Law (usually used to describe creep behavior of such materials):
  - Initial yield Norton law
  - Current yield Norton law
  - Hyperbolic sinus law (Prandtl law)
  - Power law
  - Time law

Remark: Note that the constitutive equations for the "high strain rate sensitivity" and the "creep law" models are the same for the Norton, Prandtl and power laws. The difference between the two versions is made on the characteristic values of the related parameters.

Initial yield Norton law

$$\dot{p} = \frac{\sigma_Y}{\eta}\left(\frac{f}{\sigma_Y}\right)^m \tag{6-165}$$

Current yield Norton law

$$\dot{p} = \frac{\sigma_Y}{\eta}\left(\frac{f}{\sigma_Y + R(p)}\right)^m \tag{6-166}$$

At first look, it is true to say that the two Norton laws are very similar. In the initial yield Norton law, the viscoplastic stress (*f*) depends only on the accumulated plastic strain rate and the initial yield stress (σ_y) which is constant. However, in the current yield Norton law, f depends on the accumulated plastic strain rate, the yield stress σ_y and the hardening stress *R(p)*. This difference is important because it means that the viscoplastic stress is being updated as the hardening stress increases. This second formulation is more currently used than the first one and is more recommended.

As the strain rate increases, the material stiffens in such a way that the viscoplastic behavior becomes more and more linear for both Norton laws.

Hyperbolic sinus law (Prandtl law)

$$\dot{p} = \frac{\sigma_Y}{\eta}\left(\sinh\left(\frac{f}{\beta}\right)\right)^m \tag{6-167}$$

Here, β is defined as the second viscoplastic coefficient. It has essentially the same kind of effect as the viscoplastic coefficient but to a different degree of sensibility.

As for the initial yield Norton law, the viscoplastic stress in the hyperbolic sinus law depends on the yield stress but not on the hardening stress. This law however shows a different type of strain rate dependency from the Norton laws since, instead of heading towards a linear mechanical response at high strain rates, the shape of the curve remains essentially the same as that of a quasi-static loading. That leads to say that for different strain rates, the curves all have similar nonlinear shapes but stack one above another as they reach different maximum levels of stress. It means that the hyperbolic sinus law allows to get saturation in stress even at high strain rates.

Power law

$$\dot{p} = \frac{\sigma_Y}{\eta}\left(\frac{\sigma_{eq}}{\sigma_Y + R(p)}\right)^m \tag{6-168}$$

This law has a very similar form to the current yield Norton law, the difference lying in the fact that the accumulated plastic strain rate depends upon the von Mises stress instead of the viscoplastic stress.

Remark: Up to now, this power law formulation of viscoplasticity has not demonstrated very good correlations on anisotropic viscoplastic materials using the homogenization approach of Digimat. They are also more prone to convergence issues. Therefore, these viscoplastic laws are not recommended for common use.

Time Law

$$\dot{p} = A(\sigma_{eq})^m t^n \tag{6-169}$$

The time law is a creep law which depends on time and on the von Mises stress. Three parameters are needed to completely define this creep law. The range of values for the three parameters A, m and n respectively called the creep coefficient, the creep exponent and the second creep exponent are: $A > 0$, $m > 0$ and $-1 \quad n < 0$. Contrary to the previously defined high strain rate laws, the time law does not depend on the yield stress and hardening parameters.

Strain rate dependent yield stress: Typically viscoelastic-viscoplastic materials exhibit rate-dependent yield stresses. This can be taken into account in Digimat as well, i.e., that the limit between the viscoelastic and the viscoplastic domains is rate-dependent also,

$$\sigma_y = \Upsilon(\dot{\varepsilon}) \tag{6-170}$$

Three different models are available to describe the rate dependency of the yield stress:

- **Piecewise linear**: the evolution law of the yield stress is described by a function.
- **Cowper Symonds model**: the yield stress is related to the strain rate via the Cowper-Symonds law

$$\sigma_y(\dot{\varepsilon}) = \sigma_{y,0}\left[1 + \left(\frac{\dot{\varepsilon}_{eq}}{\dot{\varepsilon}_0}\right)^{1/q}\right] \tag{6-171}$$

- **Cowper Symonds Log model**: the yield stress is related to the logarithm of the strain rate through:

$$\sigma_y(\dot{\varepsilon}) = \sigma_{y,0}\left[1 + \left(\log\frac{\dot{\varepsilon}_{eq}}{\dot{\varepsilon}_0}\right)^{1/q}\right] \tag{6-172}$$

The three parameters respectively define the initial yield stress, i.e., the yield stress in quasi-static conditions, the strictly positive Cowper-Symonds exponent and strain rate.

The total strain rate is a scalar computed as the norm of the rate of the total strain tensor as follows:

$$\dot{\varepsilon}_{eq} = \sqrt{\frac{2}{3}\dot{\varepsilon} : \dot{\varepsilon}} \tag{6-173}$$

The different models should enable a good fit of experimental data, since it is usually observed that the stress σ_y separating VE and VP domains increases with strain rate.

Units

Digimat-MF is not bound to a particular system of units. Nevertheless, units must be consistent throughout the definition of the material model, e.g., if the shear modulus is given in MPa, so should be the bulk modulus, and similarly for other dimensional parameters of the model.

Table 6-15 lists all the parameters introduced in this page, their denomination as well as their corresponding dimensions.

In addition, if the created material is to be used in a coupled simulation with a finite element code, the user will make sure that the chosen unit system is consistent with the one used in the finite element model.

Table 6-15 Parameter names, symbols, dimensions and SI units

Name	Symbol	Dimensions - M, L, T, θ, I	SI units
Density	ρ	ML^{-3}	kg/m^3
Shear modulus	G	$ML^{-1}T^{-2}$	$Pa = N/m^2$
Bulk modulus	K	$ML^{-1}T^{-2}$	$Pa = N/m^2$
Prony weight	w_i	1	-
Relaxation time	τ_i	T	s
Yield Stress	σ_Y	$ML^{-1}T^{-2}$	$Pa = N/m^2$
Initial yield stress	$\sigma_{Y,0}$	$ML^{-1}T^{-2}$	$Pa = N/m^2$
Cowper-Symonds exponent	q	1	-
Reference strain rate	$\dot{\varepsilon}_0$	T^{-1}	s^{-1}
Hardening modulus	R_∞	$ML^{-1}T^{-2}$	$Pa = N/m^2$
Hardening exponent	n	1	-
Hardening modulus 2	k	$ML^{-1}T^{-2}$	$Pa = N/m^2$
Creep coefficient	η	$ML^{-1}T^{-2}$	$Pa.s = N/m^2.s$
Creep exponent	m	1	-
Creep coefficient 2	β	$ML^{-1}T^{-2}$	$Pa = N/m^2$

Strain Rate Elasto-plasticity

It is commonly observed that the loading rate affects the mechanical response of some materials, especially in the field of plastics and composites. This is caused by the so-called viscous effects. These can be decomposed more specifically into viscoelastic and viscoplastic effects. A variation of the strain rate does in such case result in a change on the apparent stiffness of the material.

Taking into account the strain-rate dependency in the mechanical behavior law is of great importance when addressing many industrial problems like impact tests, crash tests, high rotational velocity loadings,... The objective of this documentation section is to describe how Digimat addresses these viscous effects as much as the models that are available to describe them. Some guidelines and limitations will also be proposed to help you define the parameters of the strain rate elastoplastic (SREP) material model.

Compared to conventional VE-VP materials, strain rate elastoplastic material model tackles the viscous effects by defining an equivalent elastoplastic material with strain rate dependent material parameters functions. Such approach takes advantage of the second order homogenization method, which enabled it to circumvent the limitation of the VE-VP materials. The strain rate dependence can be independently imposed to the Young's modulus, the Poisson's ratio, the yield stress, the hardening parameters or the plastic strain multiplier. The strain rate used to reconstruct the equivalent elastoplastic material is given by the imposed loading strain rate. Such definition constrains the present SREP material to constant strain rate loadings in Digimat-MF. The material model will take its full power when used in Digimat-CAE with the Hybrid solution. During the generation of the hybrid parameters, the strain rate dependent material will be transformed into a VEVP material model with Prony Series and viscous function which enables its usage for any loadings.

The homogenization of a composite with SREP phase is based on the same homogenization scheme as composite with EP phase (see section Rate-independent Inelastic Composites and Second-order Homogenization). For the homogenization procedure, the modified spectral method is imposed as isotropic extraction method (see chapter Isotropic Extraction Methods) in combination with the exponential and linear hardening model.

Thermo-strain Rate Elasto-plasticity

It is commonly observed that the mechanical and thermal loading rate affect the mechanical response of some materials, especially in the field of plastics and composites. This is caused by the so-called viscous effects. These can be decomposed more specifically into visco-thermoelastic and visco-thermoplastic effects.

A variation of the strain rate or of the temperature does in such case result in a change on the apparent stiffness of the material. The thermo-strain rate elastoplastic (TSREP) material model is the only material model that takes temperature and strain rate dependency on both the linear and non-linear behavior into account. This is done through the usage of N-dimensional dependencies (see N-dimensional Dependent Mechanical Properties).

Taking into account the strain-rate dependency in the mechanical behavior law and the temperature effects is of great importance when addressing many industrial problems like impact tests, crash tests, high rotational velocity loadings, creep tests, etc... The objective of this documentation section is to describe how Digimat addresses these viscous effects as much as the models that are available to describe them. Some guidelines and limitations will also be proposed to help you define the parameters of the TSREP material model.

Taking advantage of the SREP formulation (see section Strain Rate Elasto-plasticity), TSREP material model tackles the viscous effects by defining an equivalent elastoplastic material with temperature and strain rate dependent material parameters functions. Such approach takes advantage of the second order homogenization method which is the best in class homogenization method.

The temperature and strain rate dependencies can be independently imposed to the Young's modulus, the Poisson's ratio, the yield stress, the hardening parameters or the plastic strain multiplier. This requires the definition of a 2D function where X1 stands for the strain rate variable (interpolated as a piecewise log-linear function), and X2 stands for the temperature (interpolated as a piecewise linear function). See figures 6-10 and 6-11 for an example of assignation. The thermal expansion coefficient is restricted to temperature dependency (using a 1D function).

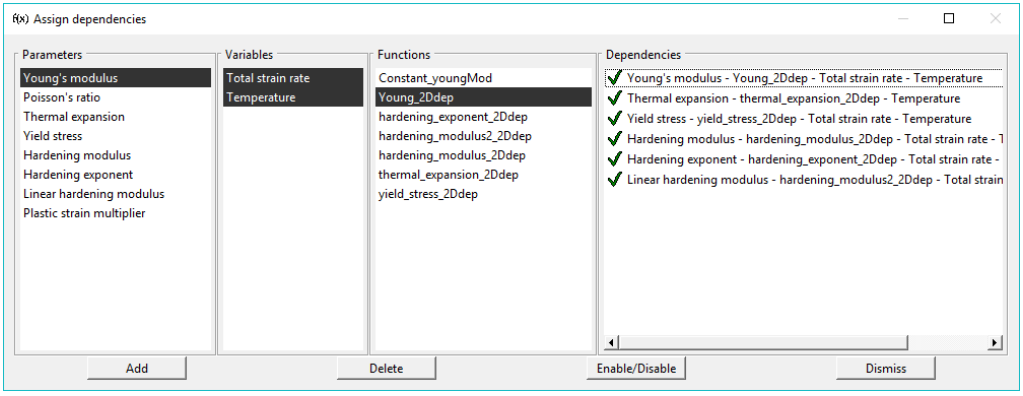


Figure 6-10 Defining N-dimensional dependencies for TSREP material parameters.

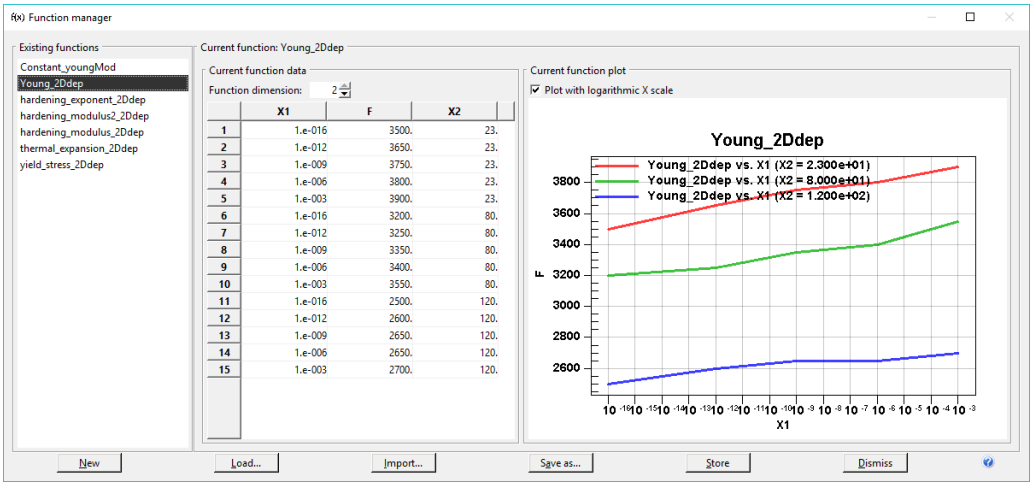


Figure 6-11 Defining N-dimensional dependencies for TSREP material parameters.

The thermal expansion coefficient is restricted to temperature dependency. The strain rate used to reconstruct the equivalent elastoplastic material is given by the imposed loading strain rate. Such definition constrains the present TSREP material to constant strain rate loadings in Digimat-MF. The material model will take its full power when used in Digimat-CAE with the Hybrid solution. During the generation of the hybrid parameters, the temperature and strain rate dependent material will be transformed into a TVEVP material model with Prony Series, a shift function and viscous function which enables its usage for any loadings.

The homogenization of a composite with TSREP phase is based on the same homogenization scheme as composite with EP phase (see section Rate-independent Inelastic Composites and Second-order Homogenization). For the homogenization procedure, the modified spectral method is imposed as isotropic extraction method (see Chapter 5: Isotropic Extraction Methods) in combination with the exponential and linear hardening model.

(Thermo-)hyperelasticity

Hyperelastic materials are materials which can be subjected to very high levels of deformation without exhibiting permanent deformations after unloading. These materials can experience incompressible or (quasi-) incompressible behaviors and are widely used in the industry, e.g., tires, seals, anti-vibration systems and so on. Hyperelastic models have been developed to model such nonlinear stress-strain relations as the ones shown by these materials.

Five hyperelastic material models are implemented in Digimat-MF. They are presented in this section. Firstly the theoretical aspects of the modeling of (thermo-)hyperelastic materials, accounting for incompressibility and large strains are presented. The strain energy function for each hyperelastic model implemented in Digimat- MF is described as well as the incompressibility methods available to handle (quasi) incompressibility. Secondly an example of the modeling of hyperelastic materials is given. Finally the major limitations relative to the use of hyperelastic models and some guidelines are presented.

Theoretical Introduction

General Introduction to Finite Strains Continuum Mechanics

We consider a body which occupies, at time t $= 0$, a configuration Ω_0 - called reference - and, at time t > 0, a configuration Ω_t - called current (see Figure 6-12).

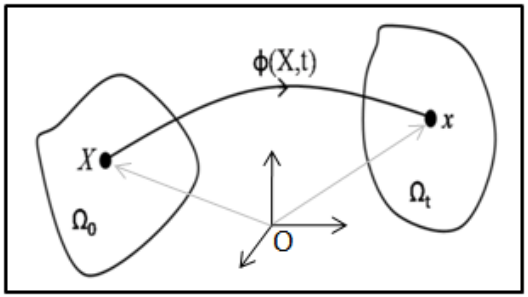


Figure 6-12 Position vectors of a material particle are X in the reference configuration and $X = \varphi(X, t)$ in the current configuration.

The motion of the body is defined by the transformation

$$x = \phi(X, t) \tag{6-174}$$

where x and X are the position vectors of a material particle in the current and in the reference configurations.

From this, the deformation gradient F is defined

$$F = \frac{\partial \phi}{\partial X} \tag{6-175}$$

Since the transformation from the reference configuration to the current one is given by the sum of the reference position vector and the displacement field, the deformation gradient also reads

$$F = 1 + \frac{\partial u}{\partial X} \tag{6-176}$$

where 1 denotes the second order identity tensor.

The ratio of an elementary volume between the current configuration dv and the reference configuration dV is denoted J and is defined as follows

$$J = \det F = \left(\frac{du}{dv}\right), \; J > 0 \tag{6-177}$$

Note that, from a mathematical point of view, J is also defined as the determinant of the deformation gradient F. J must be strictly positive in order to have positive volumes. For incompressible materials, $J = 1$.

The right and the left Cauchy-Green strain tensors are defined from the deformation gradient respectively by

$$C = F^T . F \text{ and } b = F . F^T \tag{6-178}$$

where F^T refers to the transpose of F. The three invariants of the right Cauchy-Green strain tensor are defined by the following expression:

$$I_1 = \mathrm{Tr}(\mathrm{C}),\ I_2 = \frac{1}{2}[I_1^2 - \mathrm{Tr}(\mathrm{C}^2)] \text{ and } I_3 = J^2 \tag{6-179}$$

The eigenvalues of the right Cauchy-Green strain tensor are $\lambda^2{}_i$ for i = 1, 2, 3. These are also the eigenvalues of the left Cauchy-Green strain tensor. The principal stretches are the square root of $\lambda^2{}_i$.

The Green-Lagrange strain tensor is defined from the right Cauchy-Green strain tensor as follows

$$\mathrm{E} = \frac{1}{2}(\mathrm{C} - 1) \tag{6-180}$$

while the nominal strain tensor is given by the polar decomposition of the deformation gradient $\mathrm{F} = \mathrm{V} \cdot \mathrm{R}$, R being the rotation tensor,

$$\mathrm{E}^{\mathrm{n}} = \mathrm{V} - 1 \tag{6-181}$$

Assuming the existence of a strain energy function W(F) for a hyperelastic material, the nominal stress tensor can be defined as

$$\mathrm{P}^{\mathrm{n}} - \frac{\partial W(\mathrm{F})}{\partial \mathrm{F}} \tag{6-182}$$

The nominal stress tensor is related to Cauchy stress tensor σ through the inverse of the deformation gradient,

$$\mathrm{P}^{\mathrm{n}} = J\mathrm{F}^{-1} \,.\, \sigma \tag{6-183}$$

It is found that the nominal stress tensor is not symmetric contrary to the Cauchy stress tensor.

Note that the first Piola-Kirchoff stress tensor P is defined as the transpose of the nominal stress tensor by

$\mathrm{P} = (\mathrm{P}^{\mathrm{n}})^{\mathrm{T}}$ and

the second Piola-Kirchhoff stress tensor S is defined as

$\mathrm{S} = \mathrm{J}\ \mathrm{F}^{-1} \cdot \sigma \cdot \mathrm{F}^{-\mathrm{T}}$.

Using the time derivative of the nominal stress tensor, it is possible to define a tangent operator which is used in the homogenization procedure

$$\dot{\mathrm{P}} = \mathrm{A} : \dot{\mathrm{F}}, \text{ with } \mathrm{A} = \frac{\partial}{\partial \mathrm{F}}\left(\frac{\partial W(\mathrm{F})}{\partial \mathrm{F}}\right) \tag{6-184}$$

The tangent operator A is a fourth-order tensor which has the major (diagonal) but not the minor symmetries. The parameters defining it are discussed later in the text.

Hyperelastic Laws Implemented in Digimat-MF

The strain energy function W(F) for each hyperelastic law implemented in Digimat-MF is a function of the principal invariants of the right Cauchy-Green strain tensor or of the principal stretches. These, for rubber-like isotropic material models, are usually scaled, following a decomposition of the deformation gradient into a volumetric and an isochoric parts

$$\mathrm{F} = \mathrm{F}^{vol} . \bar{\mathrm{F}}, \text{ with } \mathrm{F}^{vol} = J^{1/3}1, \bar{\mathrm{F}} = J^{-1/3}\mathrm{F} \tag{6-185}$$

By construction, we have

$$\det\bar{\mathrm{F}} = 1 \text{ and } \det\mathrm{F}^{vol} = J = \det\mathrm{F} \tag{6-186}$$

The isochoric part of the right and left Cauchy-Green strain tensors are defined as

$$\bar{\mathrm{C}} - \bar{\mathrm{F}}^{T} . \bar{\mathrm{F}} \text{ and } \bar{\mathrm{b}} - \bar{\mathrm{F}} . \bar{\mathrm{F}}^{T} \tag{6-187}$$

The principal invariants of these tensors are denoted as follows

$$\bar{I}_1, \bar{I}_2 \text{ and } \bar{I}_3 = 1 \tag{6-188}$$

while the eigenvalues read

$$\bar{\lambda}_i^2, \ i = 1, 2, 3 \text{ and } \bar{\lambda}_1\bar{\lambda}_2\bar{\lambda}_3 = 1 \tag{6-189}$$

By definition, the following relations between the total or isochoric principal invariants or stretches are found

$$\bar{I}_1 = J^{-2/3}I_1, \ \bar{I}_2 = J^{-4/3}I_2 \text{ and } \bar{\lambda}_1 = J^{-1/3}\lambda_i \tag{6-190}$$

The available hyperelastic formulations implemented in Digimat-MF are the following ones.

- neo-Hookean model:

$$W(\mathrm{F}) = \frac{G}{2}(\bar{I}_1 - 3) + \frac{K}{2}(J-1)^2 \tag{6-191}$$

  where G, K > 0 respectively denote the shear and the bulk moduli.

- Mooney-Rivlin model:

$$W(\mathrm{F}) = C_{10}(\bar{I}_1 - 3) + C_{01}(\bar{I}_2 - 3) + \frac{\alpha}{2}(J-1)^2 \tag{6-192}$$

  where C_{10} and C_{01} are called respectively the first and the second moduli. No particular restriction is made on these two moduli, except that the initial shear modulus of this model must be strictly positive,

  i.e., $G = 2(C_{10} + C_{0101}) > 0$.

- Swanson model:

$$W(F) = \frac{3}{2}\frac{A_1}{P_1+1}\left(\frac{\bar{I}_1}{3}-1\right)^{P_1+1} + \frac{3}{2}\frac{B_1}{Q_1+1}\left(\frac{\bar{I}_2}{3}-1\right)^{Q_1+1} + \frac{3}{2}\frac{C_1}{R_1+1}\left(\frac{\bar{I}_3}{3}-1\right)^{R_1+1} + \frac{\alpha}{2}(J-1)^2 \quad (6\text{-}193)$$

where A1, B1, C1 $>$ 0 respectively are the first, second and third moduli and P1, Q1, R1 0 the first, second and third exponents. The three moduli must be such that the initial shear modulus of this model is strictly positive.

- Ogden model:

$$W(F) = \sum_{i=1}^{N}\frac{2\mu_i}{m_1^2}\left[\bar{\lambda}_1^{m_i} + \bar{\lambda}_2^{m_i} + \bar{\lambda}_3^{m_i} - 3\right] + \frac{\alpha}{2}(J-1)^2, \quad (6\text{-}194)$$

whereμ_i refers to a modulus and m_i to an exponent, while $1 \le N \le 3$ denotes the number of pairs (μ_i, mi). No particular restriction is made on the values of the moduli and the exponents, except that the initial shear modulus of the model must be strictly positive, i.e., the sum of the μ_i must be positive for the considered number of pairs.

- Störakers model:

$$W(F) = \sum_{i=1}^{N}\frac{2\mu_i}{m_1^2}\left[\lambda_1^{m_i} + \lambda_2^{m_i} + \lambda_3^{m_i} - 3 + \frac{1}{\beta_1}\left(J^{-m_i\beta_1} - 1\right)\right], \quad (6\text{-}195)$$

where μ_i refers to a modulus, mi to the first exponent and β_i, to the second exponent which determines the degree of compressibility of the material and is related to the Poisson's ratio by

$$\beta_1 = \frac{v_i}{1-2v_i}, \quad (6\text{-}196)$$

$1 \le N \le 3$ denotes the number of triplets (μ_i, mi, β_i). No particular restriction is made on the moduli and the exponents, except that the initial shear modulus of the model must be strictly positive, i.e., the sum of the μ_i must be positive for the considered number of pairs.

Except for Storakers model, the parameter α which defines the degree of compressibility of the material, depends on the shear modulus G of the material model, the penalty factor α_p and a parameter α_d. It is defined by

$$\alpha = 1000\ G\ \alpha_p\ \alpha_d$$

The penalty factor must be strictly positive and its default value is 1.0. The value α_d depends on the degree of compressibility of the material, whether it is used in a composite material or not, and is determined by Digimat in the following way.

- If the material is homogeneous:
  - Compressible material: $\alpha_d = 0.002$
  - Quasi-compressible and incompressible: $\alpha_d = 1.0$
- If the material is defined as a phase in a composite RVE:
  - Compressible phase: $\alpha_d = 0.002$
  - Quasi-compressible and incompressible: $\alpha_d = 0.1$

Note that for the particular case of the neo-Hookean material, $\alpha = K$ if the material is incompressible. If the material is compressible, the bulk modulus must be defined by the user.

Incompressibility Methods

To deal with both the compressibility and (quasi-)incompressibility of hyperelastic materials a mixed formulation was proposed by Simo and Taylor (1991). This formulation uses four independent fields: the motion field $\phi(X,t)$, the change of volume $\theta(X,t)$, the pressure field $p(X,t)$ and a Lagrange multiplier $\lambda(X,t)$ to enforce the incompressibility constraint. This four-field energy functional, corresponding to an augmented Lagrangian formulation and defined with respect to the reference configuration, is given by

$$L(\phi, \theta, p, \lambda) = \int_{\Omega_0} [W(\bar{F}) + p(J - \theta) + \lambda(\theta - 1)]dV - \Pi_{ext}(\phi) \tag{6-197}$$

where,

$$\bar{F}(\phi, \theta) = \left[\frac{\theta}{J}\right]^{1/3} F, det\,\bar{F} = \theta \tag{6-198}$$

and J refers to the determinant of F.

Two methods are available in Digimat-MF to handle (quasi-)incompressibility of hyperelastic material:

- Penalty method (default method): the Lagrange multiplier is set to zero. The incompressibility of the material is enforced by increasing the penalty factor p in order to increase the value of the parameter. For example, if the penalty factor is equal to 1, the corresponding Poisson's ratio is equal to 0.4995 when a homogeneous material is considered or 0.495, in the corresponding phase of the composite material.
- The augmented Lagrangian method: when this method is used to enforce the incompressibility constraint of the material, a Lagrange multiplier is introduced. The incompressibility constraint is satisfied when the following inequality is true: $|\theta - 1| \leq 10^{-4}$ (default value), where θ designates the volume change.

Some Parameters for the Tangent Operator A

The tangent operator A, relating the variation of the deformation gradient to the variation of the nominal stress tensor, is derived from the four-field energy functional presented above and is a function of several parameters:

- Geometric stiffness terms: these terms do not have a strong effect on the response of the composite material but affect the convergence of the finite element code during a coupled analysis. See the Guidelines section for some recommendations about the use of these terms.
- Computation of tangent operator (mixed formulation) parameter: this refers to the volume change parameter which is only relevant for the computation of the tangent operator A during the homogenization procedure and when the material phases are (quasi-)incompressible. The volume change parameter is defined as

$$\eta = \frac{\dot{\theta}}{\dot{J}} \tag{6-199}$$

where θ measures the volume change of the phase and where J is the determinant of the average deformation gradient in the phase. Three methods are available to compute the volume change parameters, an explicit one, an incremental one and an implicit one.

$$\rightarrow \eta = \frac{\theta(t_n)}{J(t_n)},$$

$$\rightarrow \eta = \frac{\theta(t_n + 1) - \theta(t_n)}{J(t_n + 1) - J(t_n)},$$

$$\rightarrow \eta = \frac{\theta(t_n - 1)}{J(t_n + 1)} \tag{6-200}$$

These parameters have an influence on the convergence of an analysis, in the homogenization procedure, especially for (quasi-)incompressible materials. Indeed, for compressible materials, no restriction on the volume variation between the reference and the current configuration is done. This means that $\theta = J$, thus $\eta = 1$.

See also the Guidelines section for some recommendations about the use of these three methods.

Thermo-hyperelastic Formulation

In order to take into account for thermal effect, a multiplicative decomposition of the deformation gradient in an elastic and a thermal part is

$$\mathbf{F} = \mathbf{F}^e\mathbf{F}^{th}, \text{ with } \mathbf{F}^{th} = (1 + \alpha^{th}\Delta T)\mathbf{1} \tag{6-201}$$

where α^{th} refers to the coefficient of thermal expansion (not to be confused with entities related to the penalty method) of an isotropic material and ΔT, to the difference between the actual temperature T and the reference temperature T_0, i.e., the temperature for which the thermal strain of the material is zero. The strain energy of the hyperelastic model is evaluated through the elastic part of the deformation gradient.

All parameters defining the strain energy function can be temperature-dependent such as for example G(T) and K(T) for the neo-Hookean model or C_{10}(T) and C_{01}(T) for the Mooney-Rivlin model.

Units

Digimat-MF is not bound to a particular system of units. Nevertheless, units must be consistent throughout the definition of the material model, e.g., if the shear modulus is given in MPa, so should be the bulk modulus, and similarly for other dimensional parameters of the model.

The Table 6-16 lists all the parameters introduced in this page, their denomination as well as their corresponding dimensions.

In addition, if the created material is to be used in a coupled simulation with a finite element code, the user will make sure that the chosen unit system is consistent with the one used in the finite element model.

Table 6-16 Parameter names, symbols, dimensions and SI units

Name	Symbol	Dimensions - M, L, T, θ, I	SI units
Shear modulus	G	$ML^{-1}T^{-2}$	$Pa = N/m^2$
Bulk modulus	K	$ML^{-1}T^{-2}$	$Pa = N/m^2$
First moduli (neo-Hookean)	C_{10}	$ML^{-1}T^{-2}$	$Pa = N/m^2$
Second moduli (neo-Hookean)	C_{01}	$ML^{-1}T^{-2}$	$Pa = N/m^2$
First, second and third moduli (Swanson)	A_1, B_1, C_1	$ML^{-1}T^{-2}$	$Pa = N/m^2$
First, second and third exponent (Swanson)	P_1, Q_1, R_1	1	-
Shear modulus (Ogden and Störakers model)	μ_i	$ML^{-1}T^{-2}$	$Pa = N/m^2$

Table 6-16 Parameter names, symbols, dimensions and SI units

Name	Symbol	Dimensions - M, L, T, θ, I	SI units
First exponent (Ogden and Störakers model)	m_i	1	-
Second exponent (Störakers model)	β_i	1	-
Incompressibility parameter	α	$ML^{-1}T^{-2}$	$Pa = N/m^2$
Coefficient of thermal expansion	α^{th}	θ^{-1}	K^{-1}

Leonov-EGP Model

The Leonov-EGP material model is a finite-strain elasto-viscoplastic model including temperature, strain rate and pressure dependence. It is able to describe the large strain behavior of some polymeric materials such as Polycarbonate (PC), Polyethylene terephtalate (PET), Polypropylene (PP). Typically, these polymers exhibit stress softening followed by stress hardening as presented in Figure 6-13. The models implemented in Digimat are variants of the monomode version of the Eindhoven Glassy Polymer model.

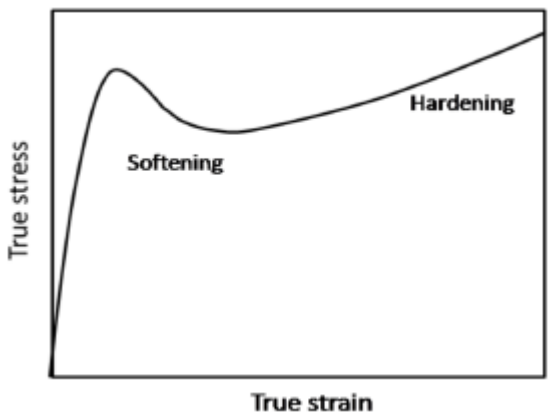


Figure 6-13 Schematic representation of the stress-strain curve for the Leonov-EGP material model.

Theoretical Introduction

The Leonov-EGP model is a finite strain elasto-viscoplastic model based on a multiplicative decomposition of the deformation gradient **F** into an elastic and a plastic part

$$F = Fe \cdot Fp. \tag{6-202}$$

As usual, in finite-strains, the elastic left Cauchy-Green strain tensor is defined by

$$\mathrm{be} = \mathrm{Fe} \cdot \mathrm{F}^{\mathrm{T}}_{\mathrm{e}} \tag{6-203}$$

with T being the transpose operator, while the plastic right Cauchy-Green strain tensor is given by

$$\mathrm{C_p} = \mathrm{F}^{\mathrm{T}}_{\mathrm{p}} \tag{6-204}$$

From a general stand point, the plastic flow for a finite-strain isotropic elasto-(visco)plastic material is defined as follows (Doghri, (2000)),

$$-\frac{1}{2}\mathrm{b}^{L}_{e}\ \mathrm{b}^{-1}_{e} = \dot{p}\frac{\delta f}{\delta \Gamma} = \dot{\varepsilon}^{p}, \text{ with } \mathrm{b}^{L}_{e} = \mathrm{F} \cdot \frac{d}{dt}(C_p^{-1}) \cdot \mathrm{F}^{T} \tag{6-205}$$

where the overhead dot and L refer to a time derivative and the objective Lie derivative respectively, p is the accumulated plastic strain, f represents the yield surface and Γ the Kirchhoff stress. The Kirchhoff stress is the Cauchy stress scaled by the determinant of the deformation gradient, $\Gamma = \mathrm{J}\ \sigma$.

Cauchy Stress Tensor Decomposition

In the Leonov-EGP model, the Cauchy stress tensor is split into a hardening stress and a driving stress, i.e.,

$$\sigma = \sigma_r + \sigma_s \tag{6-206}$$

The hardening stress can be derived from two different models which are implemented in Digimat-MF, (i) the Eindhoven model and (ii) the neo-Hookean model, respectively

$$\sigma_r = J^{-2/3}G_r b^d \text{ and } \sigma_r = J^{-5/3}G_r b^d \tag{6-207}$$

In both definitions of the hardening stress, $\mathrm{G_r}$ denotes the hardening modulus and bd, the deviatoric part of the left Cauchy-Green strain tensor $\mathrm{b} = \mathrm{F} \cdot \mathrm{F}^{\mathrm{T}}$.

The driving stress is split into a hydrostatic and a deviatoric part such as

$$\sigma_s = \sigma_s^d + \sigma_s^h \tag{6-208}$$

The hydrostatic part of the driving stress is a function of the bulk modulus K

$$\sigma_s^h = K(J-1)1 \tag{6-209}$$

where 1 denotes the second order identity tensor. The deviatoric part of the driving stress is defined as

$$\sigma_s^d = GJ^{-2/3}b_e^d$$

where G refers to the shear modulus and b_e^d, to the deviatoric part of the elastic left Cauchy-Green strain tensor b_e and J to the determinant of the gradient of deformation.

Viscosity Function

For the Leonov-EGP model, the plastic flow rule is related to the von Mises norm of the deviatoric part of the driving Cauchy stress tensor as follows

$$(\sigma_s^d)_{eq} = 3\eta(T, \sigma_m, \sigma_{eq}, S)\dot{p} \tag{6-210}$$

where the von Mises stress has the following definition

$$\sigma_{eq} = \sqrt{\frac{1}{2}\sigma_s^d : \sigma_s^d} \tag{6-211}$$

The viscosity function depends on the temperature T, the hydrostatic pressure σ_m, the rate of the accumulated plastic strain p and the state parameters S. It is given by

$$\begin{cases} \eta(T, \sigma_m, \sigma_{eq}, S) = \eta_{0,r}(T)\exp\left[\dfrac{\mu\sigma_m}{\Gamma_0}\right]\dfrac{\sigma_{eq}/\Gamma_0}{\sinh(\sigma_{eq}/\Gamma_0)}\exp(S) \\ S(t, T, \gamma_p) = S_a(t, T)R_p(\gamma_p) \end{cases} \tag{6-212}$$

$$\text{where, } \begin{cases} R_p(\gamma_p) = \left[\dfrac{1-(r_0\exp(\gamma_p))^{r_1}}{1+(r_0)^{r1}}\right]^{\frac{r_2-1}{r_2}} \\ S_a(t, T) = c_0 + c_1\log\left(\dfrac{t_{eff}+t_a}{t_0}\right) \end{cases} \tag{6-213}$$

where $\eta_{0,r}(T)$ denotes the rejuvenated viscosity coefficient which is temperature-dependent, Γ_0 the characteristic stress and μ the pressure-dependency coefficient. In this definition of the viscosity function, the accumulated plastic strain is defined as a function of the plastic strain rate as follows:

$$\dot{p} = \sqrt{\frac{2}{3}\dot{\varepsilon}^p : \dot{\varepsilon}^p} = \frac{1}{\sqrt{3}}\dot{\gamma}^p \tag{6-214}$$

The state parameter $S(t,T,\Upsilon^p)$ is split into two main contributions which act independently, the softening and aging kinetics respectively.

- $R_p(\Upsilon_p)$ represents the softening kinetics, which is a function of the accumulated plastic strain p and three material coefficients: r_0, r_1 and r_2.

- $S_a(t,T)$ represents the aging kinetics, which is a function of four materials parameters: c_0, c_1, the initial age t_a and the reference time t_0 plus the effective time t_{eff}. The latter is given by

$$t_{eff} = \int_0^t \frac{d\xi}{\alpha_T(T(\xi))a_\sigma(\sigma_{eq}(\xi))} \text{ with } \begin{cases} \alpha_T(T) = \exp\left[\frac{\Delta U_a}{R}\left(\frac{1}{T} - \frac{1}{T_{ref}}\right)\right] \\ a_\sigma = \frac{\sigma_{eq}/\Gamma_a}{\sinh(\sigma_{eq}/\Gamma_a)} \quad \text{with,} \quad \Gamma_a = \frac{RT}{\nu_a} \end{cases} \tag{6-215}$$

where R = 8.314472 J.K^{-1}.mol^{-1} refers to the universal gas constant, ΔU_a the activation energy, T_{ref} the reference temperature, T the testing temperature and ν_a a an aging activation volume. ΔU_a and ν_a are two additional material parameters.

In the Leonov-EGP model, the yield stress is strain rate-dependent and temperature-dependent. It is defined as follows

$$\Gamma_y = \Gamma_0 \sinh^{-1}\left(\frac{\dot{\varepsilon}}{\dot{\varepsilon}_0^*}\right), \text{ with } \dot{\varepsilon}_0^* = \dot{\varepsilon}_0 \exp\left[\frac{-\Delta U_a}{RT}\right] \tag{6-216}$$

where the overhead dot still refers to a time derivative and a 0 subscript to a reference value. The strain rate is defined from the deformation gradient and the time derivative of the right Cauchy-Green strain tensor as follows

$$\dot{\varepsilon} = \frac{1}{2} F^{-T} \,.\, C \,.\, F^{-1} \tag{6-217}$$

Plasticity occurs only if $(\sigma_s{}^{d})_{eq}$ becomes greater than Γ_y.

Units

Digimat-MF is not bound to a particular system of units. Nevertheless, units must be consistent throughout the definition of the material model, e.g., if the shear modulus is given in MPa, so should be the bulk modulus, and similarly for other dimensional parameters of the model.

Table 6-17 lists all the parameters introduced in this page, their denomination as well as their corresponding dimensions.

In addition, if the created material is to be used in a coupled simulation with a finite element code, the user will make sure that the chosen unit system is consistent with the one used in the finite element model.

Table 6-17 Parameter names, symbols, dimensions and SI units

Name	Symbol	Dimensions - M, L, T, θ, I	SI units
Density	ρ	ML^{-3}	kg/m^3
Specific heat	c	$L^2T^{-2}\theta^{-1}$	$J/kg.K$
Thermal conductivity coefficient	k^{th}	$MLT^{-3}\theta^{-1}$	$W/m.K$
Thermal flux	Q	MT^{-3}	W/m^2
Temperature	T	θ	K
Volumetric heat supply	R	$ML^{-1}T^{-3}$	W/m^3

Fourier Model - Thermal Conductivity

This page briefly describes the heat equation and in particular Fourier's conductivity law.

Heat Conservation Equation

The first law of thermodynamics states that in a closed system, energy is conserved over time. This statement translates into the following equation:

$$\rho c \frac{dT}{dt} = -\operatorname{div}(q) + r \tag{6-218}$$

where ρ, c, T, t, q and r respectively stand for the density, the specific heat, the temperature, the time, the heat flux and the volumetric heat supply. If the sole thermal conductivity is considered, then the heat flux can be rewritten, according to Fourier's law, as

$$q = -k^{th} \cdot \operatorname{grad}(T) \tag{6-219}$$

with k^{th} being the thermal conductivity matrix which can be:

- isotropic

$$k^{th} = k \begin{bmatrix} 1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{bmatrix} \tag{6-220}$$

- transversely isotropic

$$k^{th} = \begin{bmatrix} k_l & 0 & 0 \\ 0 & k_t & 0 \\ 0 & 0 & k_t \end{bmatrix} \quad (6\text{-}221)$$

- orthotropic

$$k^{th} = \begin{bmatrix} k_1 & 0 & 0 \\ 0 & k_2 & 0 \\ 0 & 0 & k_3 \end{bmatrix} \quad (6\text{-}222)$$

This law establishes proportionality between the conduction heat flux and the temperature gradient in the material.

For anisotropic composite materials, the thermal conductivity often proves to vary according to the microstructure of the composite, i.e., the fiber orientation distribution, due to the thermal conductivity difference between each phase. A thermal conductivity tensor is then used rather than a scalar coefficient to characterize the material thermal conductivity in every space direction.

To characterize the thermal behavior of a material in Digimat-MF, three inputs are required: (i) its density ρ, (ii) its specific heat c, i.e., the heat capacity per unit volume, and (iii) its thermal conductivity matrix k^{th}. In steady-state conditions, the sole thermal conductivity coefficient is required.

Each previous behavior implemented in Digimat-MF can be applied at the matrix level. For transversely isotropic and orthotropic behavior, the anisotropy system axis must be defined using local axis system. By default, the local axis system and the RVE's axis system are the same.

At the inclusion level, only isotropic and transversely isotropic behaviors are supported; orthotropic behavior is not supported. The local axis system of a transversely isotropic behavior is collinear to the local axis system of the inclusion. It is not possible for the inclusions phase to define another local axis system.

A temperature dependency may be set for the thermal conductivities and the specific heat capacities of the involved materials in the analysis (see Figure 6-14). The choice of such dependencies implies the use of a temperature loading that defines the temperature range to which the constituent materials will be exposed during the analysis (see Figure 6-15). The solver deduces then the corresponding temperature at each time increment and its corresponding material property value. The latter is computed by using the evolution function of the material property as function of the temperature.

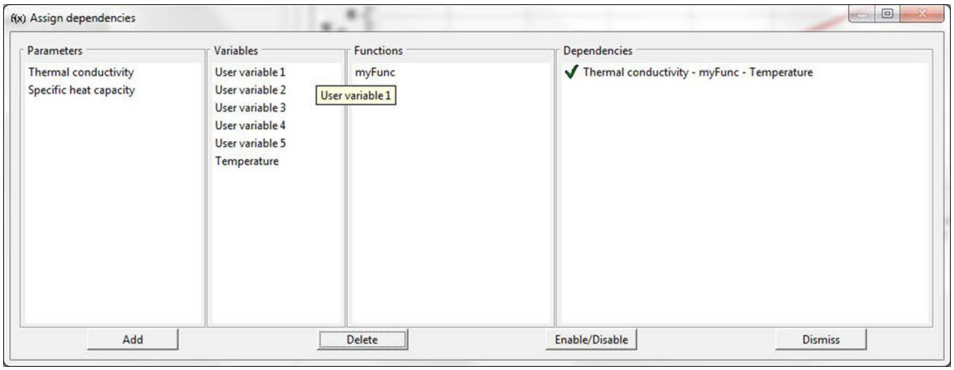


Figure 6-14 Temperature dependency of the thermal properties.

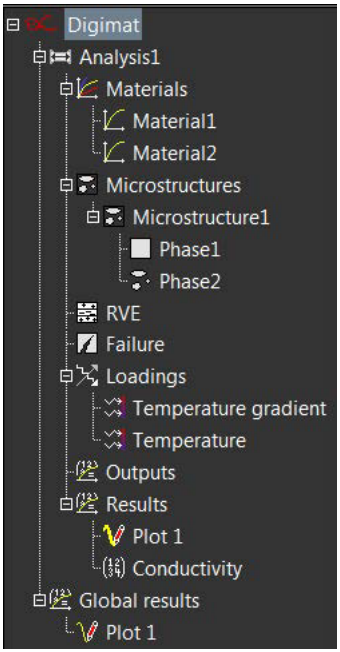


Figure 6-15 Temperature loading in a thermal analysis.

Units

Digimat-MF is not bound to a particular system of units. Nevertheless, units must be consistent throughout the definition of the material model, e.g., if the thermal conductivity is given in (W/m.K), so should be the specific heat should be in J/kg.K.

Table 6-18 lists all the parameters introduced in this page, their denomination as well as their corresponding dimensions.

In addition, if the created material is to be used in a coupled simulation with a finite element code, the user will make sure that the chosen unit system is consistent with the one used in the finite element model.

Table 6-18 Parameter names, symbols, dimensions and SI units

Name	Symbol	Dimensions - M, L, T, θ, I	SI units
Shear modulus	G	$ML^{-1}T^{-2}$	$Pa = N/m^2$
Bulk modulus	K	$ML^{-1}T^{-2}$	$Pa = N/m^2$
Hardening modulus	G_r	$ML^{-1}T^{-2}$	$Pa = N/m^2$
Rejuvenated viscosity coefficient	$\eta_{0,r}$	$ML^{-1}T^{-2}$	$Pa.s = N/m^2$
Characteristic stress	τ_0	$ML^{-1}T^{-2}$	$Pa = N/m^2$
Pressure-dependent coefficient	μ	1	-
Coefficients for softening kinetics	r_0, r_1 and r_2	1	-
Coefficients for aging kinetics	c_0, c_1	1	-
Initial age	t_a	T	s
Reference time	t_0	T	s
Coefficient of Thermal Expansion	R=8.314472	θ^{-1}	$J.K^{-1}.mol^{-1}$

Ohm Model - Electrical Conductivity

This page briefly introduces the electrical conductivity equation that relates the current density to the electrical potential field.

Electrical Conductivity Equation – Ohm's Law

Similar to Fourier's law in heat conduction, Ohm's law relates the gradient of a potential to a flux, in this case, the electric potential V and the current density J,

$$J = -k^{el}\,\mathrm{grad}(V) \tag{6-223}$$

with k^{el} being the electrical conductivity matrix which can be:

- isotropic

$$k^{el} = k\begin{bmatrix}1 & 0 & 0\\ 0 & 1 & 0\\ 0 & 0 & 1\end{bmatrix} \quad (6\text{-}224)$$

- transversely isotropic

$$k^{el} = \begin{bmatrix}k_l & 0 & 0\\ 0 & k_t & 0\\ 0 & 0 & k_t\end{bmatrix} \quad (6\text{-}225)$$

- orthotropic

$$k^{el} = \begin{bmatrix}k_1 & 0 & 0\\ 0 & k_2 & 0\\ 0 & 0 & k_3\end{bmatrix} \quad (6\text{-}226)$$

This law establishes proportionality between the current density and the electric potential gradient in the material.

For anisotropic composite materials, the electrical conductivity often proves to vary according to the microstructure of the composite, i.e., the fibers orientation distribution, due to the electrical conductivity difference between each phase. An electrical conductivity tensor is then used rather than a scalar coefficient to characterize the material electrical conductivity in every space direction.

To characterize the electrical behavior of a material in Digimat-MF, two inputs are required: (i) its density ρ and (ii) its electrical conductivity kel. Note that the density is only necessary if the composite is described in terms of mass fractions.

Each previous behavior can be applied at the matrix level. For transversely isotropic and orthotropic behavior, the anisotropy system axis must be defined using local axis system. By default, the local axis system and the RVE's axis system are the same.

At the inclusion level, only isotropic and transversely isotropic behaviors are supported; orthotropic behavior is not supported. The local axis system of a transversely isotropic behavior is collinear to the local axis system of the inclusion. It is not possible for the inclusions phase to define another local axis system.

Units

Digimat-MF is not bound to a particular system of units. Nevertheless, units must be consistent throughout the definition of the material model, e.g., if the electrical conductivity is given in $1/\Omega$.m, so should be the other dimensional parameters of the model.

Table 6-19 lists all the parameters introduced in this page, their denomination as well as their corresponding units.

In addition, if the created material is to be used in a coupled simulation with a finite element code, the user will make sure that the chosen unit system is consistent with the one used in the finite element model.

Remark: Electrical conductivities can get very low values when they are expressed in SI units. For example, a typical value for the conductivity of epoxy resin is $k^{el} = 10^{-15}$ -10^{-13} $1/\ \Omega.m$. Such values lower than 10^{-12} will typically trigger error messages in Digimat. To avoid this issue, it is recommended to use another units system (like MPaSI).

Table 6-19 Parameter names, symbols, dimensions and SI units

Name	Symbol	Dimensions - M, L, T, θ, I	SI units
Density	ρ	ML^{-3}	kg/m^3
Electrical conductivity coefficient	k^{el}	$M^{-1}L^{-3}T^3I^2$	$1/\Omega.m \equiv A^2.s^3/m^3.kg$
Current density	J	$M^{-2}I$	A/m^2
Electric potential	V	$ML^2T^{-3}I^{-1}$	$V \equiv kg.m^2/A.s^3$

Coupled Thermal Mechanical Materials

These materials are a combination of the thermomechanical and Fourier material mentioned in above sections.

The thermal behavior of these materials follows the Fourier law. Density, thermal conductivity and specific heat capacity are required inputs. Even though, density and specific heat capacity are not explicitly used by MF, they are necessary if the material is used in a FEA simulation with Digimat-CAE.

The mechanical part in such materials can be thermoelastic or thermoviscoelastic. When using thermoelastic model, the symmetry can be isotropic or transversely isotropic. When using thermoviscoelastic model, only isotropic behavior is available. The viscoelastic behavior only can be described with Prony series. Supported shift function types are WLF, Piecewise linear or Arrhenius model.

Because thermal and thermomechanical parameters are included in such material, it can be coupled with different FEA analysis type as:

- Thermal analysis;
- Structural analysis with given temperature in each node or in boundary;

- Coupled thermal/thermomechanical analysis which contains a transient dynamic analysis on thermal and a thermomechanical analysis with the temperature field obtained from thermal analysis in each increment.

As an additional option, curing or crystalline model can be activated in such materials.

Curing

All the available cure kinetics models in thermomechanical type can be supported here.

- Johnston-Hubert
- Lee, Chiu, Lin
- Modified Kamal-Sourour

More detailed introduction can be found in section Curing Model.

When using thermoviscoelastic mechanical behavior, the stiffness evolution during curing can be described by both Prony series and the shift function in curing model. When using thermoelastic mechanical behavior, the stiffness evolution can be defined as a function of temperature, as explained in section Curing Law.

If the selected analysis type is coupled thermomechanical, the heat generated by the curing reaction is taken into account. Therefore, an additional material parameter must be provided, namely the total specific heat released during the complete curing reaction.

Crystalline

Double Nakamura Weibull model and Nakamura Hoffman Lauritzen model can be selected for describing the crystalline behavior. The detail description on the formulas can be found in section Crystallinity Model.

Similar to Digimat-AM, the phase changing temperature and max volumetric crystal shrinkage strain can be defined. The latent heat is described in phase changing temperature, which will not be used in MF computation but available when coupled with FEA code.

7 Microstructure

- Composite Microstructures
- Microstructure Types
- Theory

Composite Microstructures

By definition, a composite implies the combination of two or more phases. These phases can refer to the same material in different molecular configurations, but most of the times they are made out of different materials. When dealing with composites in a micromechanical approach, a good definition of the phases that constitute the whole microstructure is as important as a good definition of the materials themselves.

The microstructure definition refers to many parameters like the content of each inclusion phase, the aspect ratio, and the orientation definition of the inclusions, or the presence or not of coatings around an inclusion phase or the presence of bundle.

A Digimat analysis must absolutely have a microstructure defined. To ensure a good understanding of this definition step of a composite microstructure, this section of the documentation details all that must be defined to completely describe a microstructure.

Microstructure Types

The type of microstructure must be defined first. There are three types of microstructures in Digimat-MF:

- **Generic**: The generic type consists of a user-defined combination of matrix, inclusion, void and/or continuous fiber phases.
- **Fabric**: This type consists of a user-defined combination of matrix and yarns and requires the definition of a weave pattern or the filament cross-section for lattices.
- **Lattice**: This type consists of a user-defined definition of the cross-section of unreinforced filament.
- S**heet Molding Compound**: This type consists of a forced definition of a matrix reinforced by bundles of very long fibers modeled as continuous fibers.

Fabric Microstructure

Fabric microstructure can be defined by choosing the **Fabric** option in the main Microstructure tab item in the Digimat tree. You can choose between two families of fabrics: woven and braided. The woven choice is itself subdivided into 4 possibilities: 2D, 2.5D and 3D woven whether interlock or orthogonal (see Figure 7-1).

For all fabric microstructures, a matrix and yarn phases must be defined. If already existing, it is possible to assign the phases to the fabric microstructure. If no relevant phase exist yet, then the Definition tab of Microstructure allows to create matrix and yarn phases on-the-fly via the dedicated button **Add matrix phase** or **Create yarn phase**. Creating new phases will populate the Digimat tree under the Microstructure item.

Two type of yarn phases are available

- Basic yarn

- Advanced yarn.

If the selected microstructure contains 'Basic yarn': In this case, the woven will use a simplified modeling approach that does not take undulations into account. The material properties of the yarn will be the one of the fibers. The microstructure is solely describes by the warp angle and the warp/weft angle for woven and by the braiding angle and inlays presence for braided.

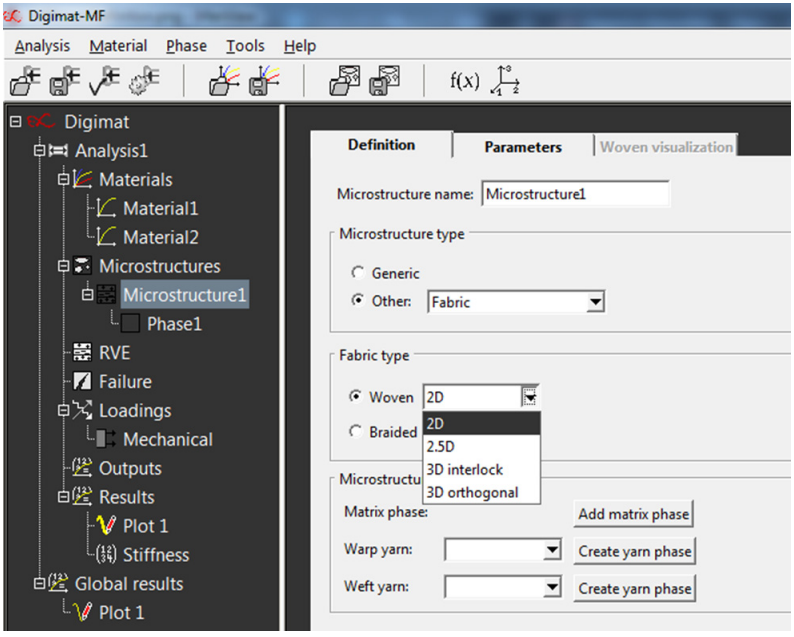


Figure 7-1 Definition of a fabric RVE.

If the selected microstructure contains 'Advanced yarn': The woven will use an advanced modeling strategy. This strategy is based on a computation of the geometry of the different yarns in the woven, in order to take into account the effect of the undulations of the yarns on the macroscopic response.

The material properties of the yarn will be computed from the material properties of the matrix and the fibers. The microstructure requires a more detailed description of the weave pattern. As Digimat-MF and Digimat-FE share the common workflow for fabric definition with advanced yarns, please refer to the subsection Fabric Microstructures in *Digimat-FE User's Guide* to review the definition of fabric microstructures with advanced yarns. The fabrics capabilities that are restricted to Digimat-FE are written in the dedicated Known limitations section RVE.

Lattice Microstructure

Another special microstructure in Digimat consists of the Lattice microstructure, which is typically encountered in additive manufacturing applications. It can be accessed via the Microstructure tab, by choosing **Other** type and selecting the **Lattice** possibility.

As the lattice capabilities differ in Digimat-MF and Digimat-FE, this section will focus solely on the Digimat- MF ones.

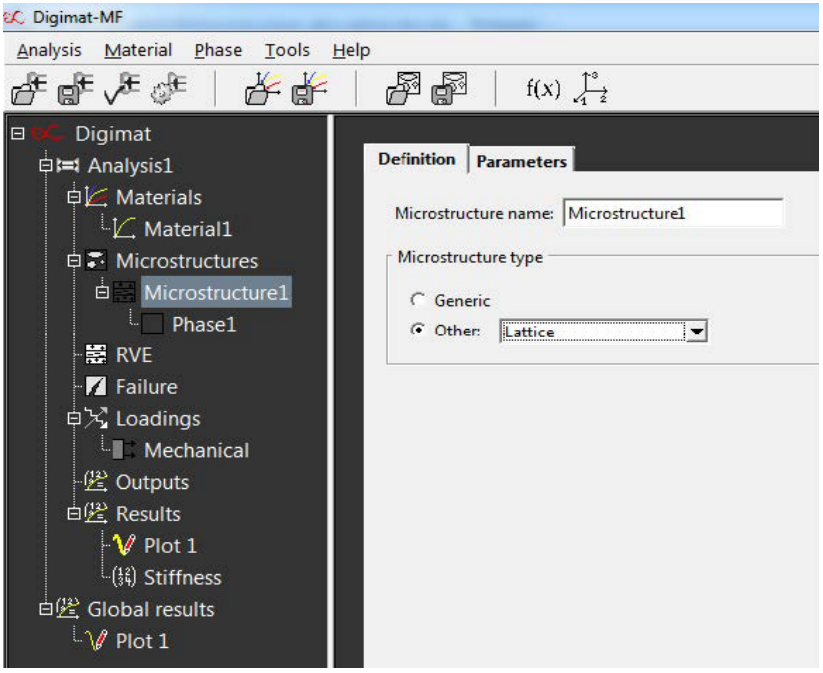


Figure 7-2 Definition of Lattice Microstructure.

Once the Lattice type is selected, the actual microstructure can be defined in the Parameters tab. Only dense filament microstructures are available. These can be characterized through the definition of:

- Ratio between extrusion width and layer height: describes the overall shape of the filament based on the ratio of its width and height
- Relative bonded width: describes the proportion of filament width which is in direct contact with its neighbour filament
- Relative bonded height: describes the proportion of filament height which is in direct contact with its neighbour filament

Once values are set for all parameters, the interbead porosity is automatically computed and the microstructure 3D visualization becomes available. The filament inputs allow to compute the amount of porosity and the porosity shape.

Only single phase lattice microstructures are supported, so only a single Matrix type of phase may be defined under a Lattice microstructure. The matrix material is limited to elastic and J_2-elastoplastic.

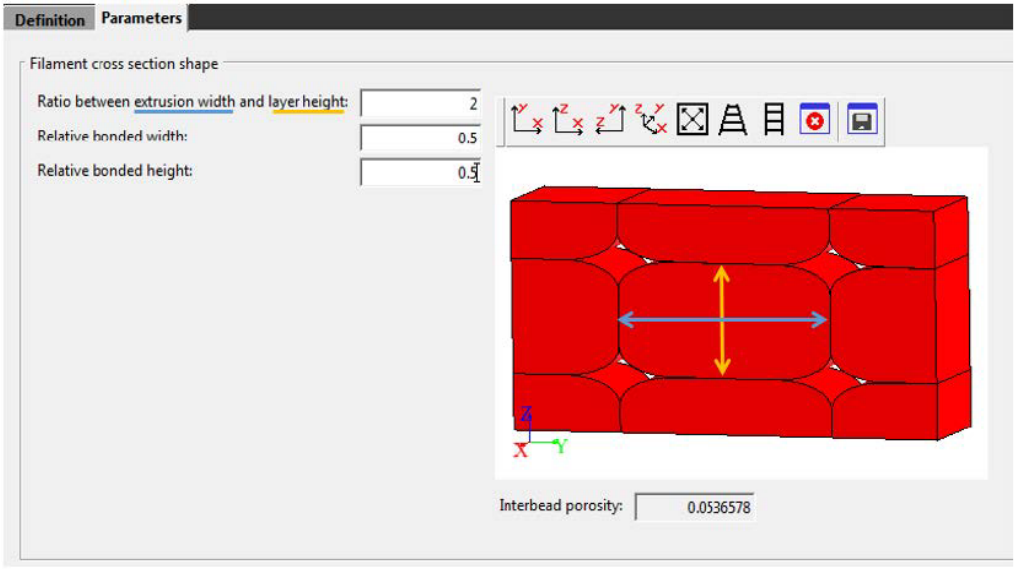


Figure 7-3 Definition of a Filament Microstructure.

Sheet Molding Compound Microstructure

This microstructure covers different type of sheet molding compound which can be modeled as matrix reinforced by bundle of very long fibers. The microstructure can only contain a matrix and a continuous fiber phase.

The material properties are restricted to elastic and viscoelastic materials.

Once the Sheet molding compound is selected, the user can define the bundle geometrical parameters as its orientation tensor. The bundle is defined by the following parameters:

- Ratio between the length and the width of the bundle
- Ratio between the width and the height of the bundle
- Fiber volume fraction in bundle

The orientation tensor definition is the same as for generic microstructure.

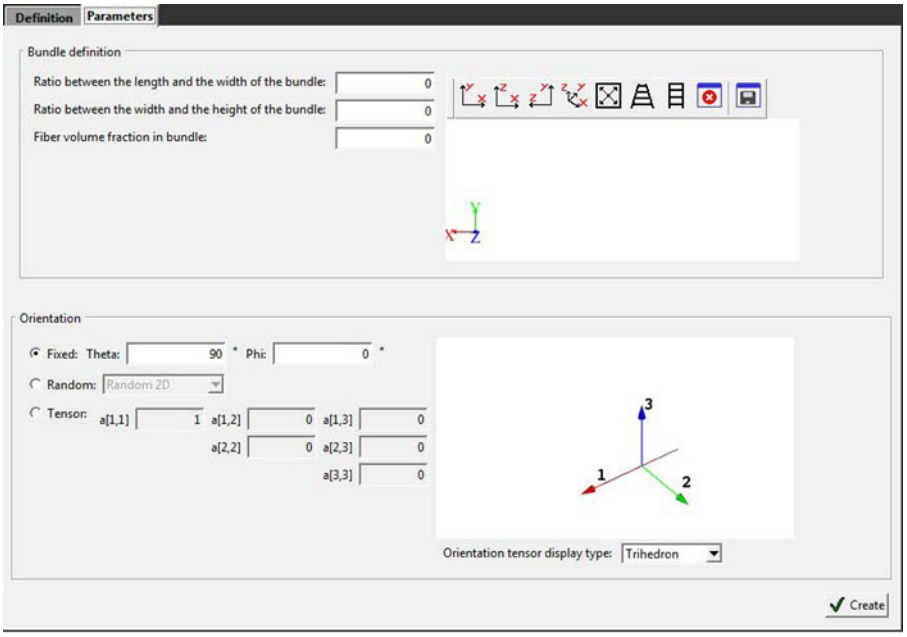


Figure 7-4 Definition of a Sheet Molding Compound microstructure.

Metal

This microstructure covers metals which can be modeled as polycrystalline aggregates. The microstructure can only contain one or two grain phases.

The material properties are restricted to elasto-plasticity and crystal-plasticity materials.

If a metal microstructure is used in combination with a crystal plasticity material model, crystallographic texture can be considered: either a random 3D texture or a user-defined texture. A single generic ASCII format (`.txt`) is supported. The input file should provide at least 3 columns, corresponding to the 3 Euler angles (Bunge convention) and eventually a fourth column corresponding to the weight of each orientation.

When an input file is provided, the texture is discretized using one of the following discretization technique:

- **Monte-Carlo** lottery
- **STAT**: statistical discretization developed by Toth (1992)
- **Hybrid-IA**: integral approximation approach developed by Eisenlohr (2008)

Theory

Matrix / Inclusion / Void microstructures

Any material defined in Digimat must always have at least one phase and must always have a matrix phase. That means if you define a material with only one phase, this material will necessarily be the matrix phase, which implies it will finally correspond to a homogeneous material. In such case, no homogenization step needs to be done by Digimat to compute its response to any loading. Notice also that no parameter needs to be defined under the matrix phase, this phase is by default assigned a deformable behavior. All one needs to do is to assign the material corresponding to this phase in the list of the defined materials and maybe give it a personalized name.

For composites made of more than one phase, all phases in addition to the first mandatory matrix phase are considered as inclusion phases; a void phase can also be added in the microstructure characterization. This is another way of saying it is possible to define as many inclusion phases as needed, but only one matrix phase can be defined.

If a microstructure contains two or more phases, the homogenization of the response of each phase to a loading is then required to compute the macro response of the composite to that same loading. This can be done either with Mori-Tanaka or with the Double inclusion model (see section MFH Models for Two-phase Composites).

If there are more than one inclusion phase in the composite, a choice is also given on how to compute the homogenization step, this choice being the multi-step and the multi-level methods (see section Multi-phase Composites).

Matrix Phase

The definition of a matrix phase in Digimat is fairly straightforward as only a material needs to be assigned to it. By default, this phase is considered as deformable.

The microstructure and phase names can be modified via the matrix Type tab.

Inclusion Phase

As for the matrix phase, the user must assign the material corresponding to the inclusion phase he defines from the list of existing materials and he may give it a personalized name. But the complete definition of an inclusion phase is not as straightforward as that of the matrix phase and there are other parameters to input, being the inclusion phase behavior, the phase content definition, the geometry of the inclusions and their orientation definition in the RVE. Notice those last three parameters are explained in the Phase Parameters section since it is common to both the inclusion and the void phases.

Behavior: Three different inclusion behaviors, referring to how the inclusions are considered to compute the mechanical response of the RVE when subjected to a loading, are available in the inclusion phase definition:

- **Deformable**: This is the default and usual behavior selected by users. It is available for any type of inclusion: elastic, elasto-plastic, elasto-viscoplastic, hyperelastic and so on. Selecting this option will trigger the use of the inclusion material model throughout the computations of the RVE response.
- **Incrementally rigid:** This behavior has been developed with the purpose of reducing computation time. In the elastic regime it works exactly the same way as for the deformable behavior. However in the plastic regime, it assumes that the inclusion phase is much stiffer than the matrix so that the strain increments in the inclusion phase can be neglected in the computation of the macroscopic strain of the RVE.

  No strain is applied to the inclusion phase and from a strain averaging relation, the strain increment of the matrix can be directly related to the macroscopic strain, as shown by the following expression:

$$\Delta\varepsilon_0 = \frac{1}{\nu_0}\Delta E \tag{7-1}$$

  where 0 denotes the matrix volume fraction, E and ε_0 the macroscopic and matrix phase strain tensors. From the matrix behavior, the stiffness tensor is known and the increment of stress on the matrix can be computed.

  Although there is no strain increment in the inclusions, they still contribute to the mechanical behavior of the RVE such that their stress tensor must be known. Since this is not yet the case, the macroscopic Cauchy stress tensor Σ must then be computed. This is done via the following equation:

$$\Sigma = C : \dot{E} \Leftrightarrow \Delta\Sigma = C : \Delta E \tag{7-2}$$

  where C denotes the macroscopic stiffness defined as follows:

$$\begin{aligned} C &= [\nu_1 C_1 : B^{\varepsilon} + (1-\nu_1)C_0] : [\nu_1 B^{\varepsilon} + (1-\nu_1)I]^{-1} \\ \text{with } \varepsilon_1 &= B^{\varepsilon} : \varepsilon_0 \end{aligned} \tag{7-3}$$

  In this last equations, ν_1 denotes the fiber volume fraction, B^{ε} is called the localization tensor and I refers to the identity matrix. The indices 1 and 0 respectively refer to the inclusion and matrix phases.

  Then, making the assumption that the fibers are infinitely rigid (i.e., infinite stiffness), the stiffness expression can be simplified in the following form:

$$C = C_0 + \left(\frac{\nu_1}{1-\nu_1}\right) P^{-1} \tag{7-4}$$

  in which P^{-1} is known and denotes the polarization vector. Knowing the stiffness tensor of the matrix phase, the macroscopic stiffness can finally be computed and from that computation, it is also possible to compute the inclusion phase stress tensor with the following equation:

$$\sigma_1 = \frac{1}{\nu_1}[\Sigma - (1 - \nu_i)\sigma_0] \quad (7\text{-}5)$$

The above described simplifications in the handling of the homogenization procedure lead to a reduced computation time. The usage of this method is recommended when the matrix material has a behavior close to perfectly plastic.

Notice that this phase behavior is restricted to elastic inclusions embedded in an elastoplastic or an elasto-viscoplastic matrix. Moreover, it is disabled for a multi-layer RVE with an elasto-viscoplastic matrix.

- **Rigid:** This rigid inclusion behavior aims at reducing computation time and stabilizing the homogenization scheme of analyses on composites made of stiff elastic fibers embedded in a soft hyperelastic matrix. The hyperelastic material is the only model supported for this inclusion formulation.

The main hypothesis of this formulation is to assume the inclusion phase as perfectly rigid (i.e., infinite stiffness) such that its contribution to the macroscopic strain of the RVE is neglected both in the elastic and in the plastic regimes. Due to this assumption, no material need to be assigned to the inclusion phase with this inclusion behavior.

Notice also that this behavior is *limited* to spherical inclusions (i.e., aspect ratio of 1) for FE computations done with coupled CAE software.

Void Phase

The void phase, as its name indicates it, refers to a capability in Digimat to model air inclusions in a composite.

When this phase is selected, the stiffness is assumed to be zero and no material needs to be assigned to it. As for the inclusion phase, some phase parameters, described later on in this section of the documentation, about content, shape and orientation of the voids, are required.

Remark: When voids are considered, the homogenization multi-level method can overcome some deficiencies of the default multi-step method. Indeed, the multi-step method may predict a plastic composite response which is stiffer with voids than without voids. Therefore the multi-level method is advised.

Continuous Fiber Phase

The continuous fiber phase is a particular case of inclusion phase. It has been simplified such that all parameters that are not relevant for continuous fibers are removed. For example, it doesn't make sense to define an orientation tensor or a random orientation or an aspect ratio for continuous fibers.

Yarn Phase

The yarn phase is meant to be used in woven microstructures. Two different types of yarn can be defined: basic yarn and advanced yarn. The basic yarn is meant to be used for basic modeling

of woven microstructures, where the weave pattern will not be taken into account in the computation of the homogenized composite properties. On the other hand, the advanced yarn is used when a more precise homogenization is desired, taking into account the effective shape of the yarns (undulations caused by the weaving process).

Phase Parameters

Once the type of phase (i.e., inclusion or void), a behavior and whether or not the inclusion phase is coated, a series of parameters need to be defined, as presented in Figure 7-5.

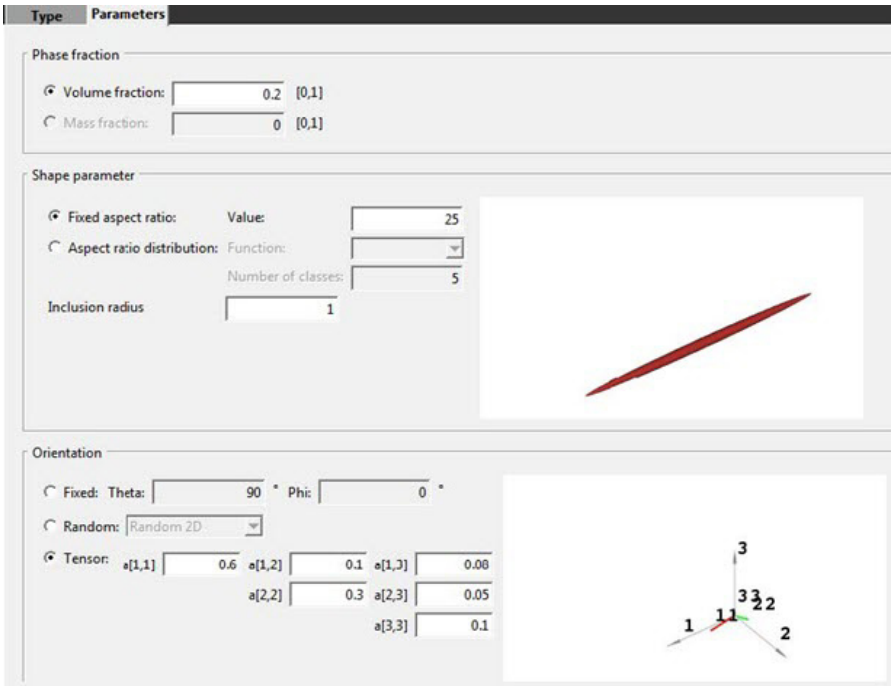


Figure 7-5 Definition of parameters for inclusion or void phases.

Content

The content of each inclusion phase must be in the range of 0 to 1 (excluded), and can be expressed in terms of mass or of volume fraction. In the same microstructure, mixing the two possibilities is however not supported.

Digimat code works only with volume fraction in every homogenization formulation. If defining mass fractions, these will automatically be converted into a volume fraction and to do so, it requires that the density of each material constituting the composite be correctly defined.

Regarding voids, their content can only be given by a volume fraction. Since mixing volume and mass fractions is not supported, that implies all inclusion phases to be also defined with volume fractions for such cases.

Specifying the phase content via the mass fraction is also impossible when the phase behavior is modeled via the rigid formulation.

Shape

The shape actually refers to the ratio of the length (L) and the diameter (D) of the inclusions/voids. There are three different ways in Digimat to define the geometry of the inclusions. A graphical interactive 3D representation of the inclusion shape is displayed.

Remark: As a matter of simplicity, notice that inclusions and voids will be referred to as inclusions only for the coming subsections.

Fixed Aspect ratio: This shape parameter defines the aspect ratio L/D of an inclusion modeled as a spheroid (i.e., an ellipsoid of revolution), where L is the length along the axis of revolution and D is the diameter in the plane orthogonal to the axis of revolution.

- A spherical inclusion has an aspect ratio of 1.
- Fibers' aspect ratio must be set to a value larger than 1, often in the range of 15 to 30 for short fibers, and much higher for long fibers.
- Laminates should have a very high aspect ratio; notice aspect ratios equal or larger than 1000 are considered as infinite in Digimat computations.
- Finally, the aspect ratio of platelets must be smaller than 1 but greater than 0 (Figure 7-6).

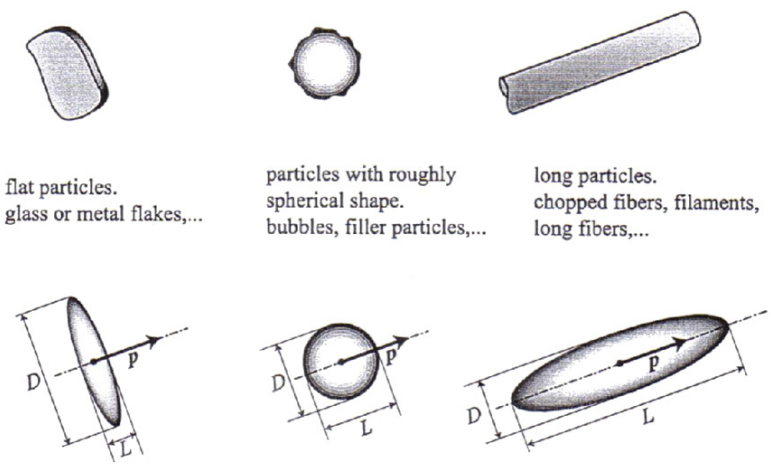


Figure 7-6 Various types of inclusions.

Figure 7-6 shows various types of inclusions with a length smaller, equal or greater than the diameter. A large number of reinforcements found in composite materials can be modeled as spheroids, including filler particles, flakes, short and long fibers Lielens (1999).

Aspect ratio distribution: The aspect ratio of an inclusion phase can also be defined through a distribution function, which can be determined from experimental observations (see Figure 7-7). The user has to first define the aspect ratio distribution function under **Tools** > **Functions**. Then in the parameters tab shown in Figure 7-5, the user must on one hand refer to this function's name by clicking on **set**, and on the other hand specify the desired **number of classes** used to discretize the aspect ratio distribution function. This number can be the exact number of data points defining the distribution, which is for instance 16 in the distribution of Figure 7-7, but it can also be any other number. If it is different from the number of data points entered in the distribution, the aspect ratio of each class, that we will call AR(i), is a linear interpolation between the two surrounding data points.

The discretized function is displayed graphically as an histogram, next to the inclusion shape display.

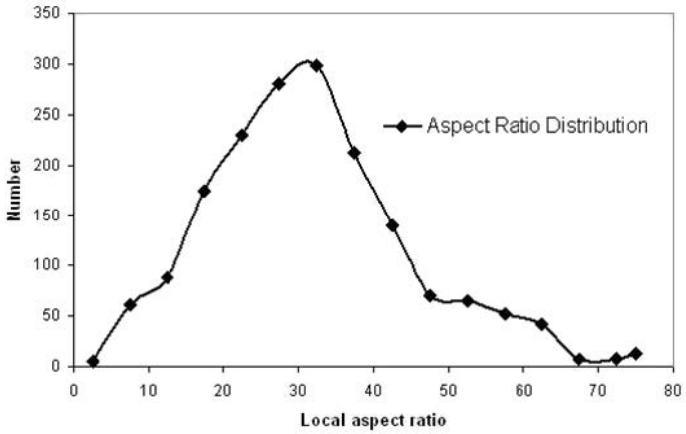


Figure 7-7 Example of an aspect ratio distribution.

Digimat automatically creates as many inclusion phases as the number of classes asked for, all having the same orientation and the same mechanical properties. Obviously, a large number of classes increases the accuracy but having more phases also increases the computation time. This is why it is recommended to seek for a compromise keeping the number of phases as low as possible.

The local fraction (volume/mass) of each inclusion phase i is computed by Digimat as follows:

$$\nu_i = \frac{AR^{(i)}N^{(i)}}{\sum_{j=1}^{n} AR^{(j)}N^{(j)}}\nu_{phase} \qquad (7\text{-}6)$$

where $N^{(j)}$ denotes the number of each local aspect ratios $AR^{(j)}$ (both values are extracted from the distribution function in Figure 7-7), n denotes the number of phases used to discretize the distribution function, and Phase is the global fraction of the overall inclusion phase.

Remark: Only one inclusion phase per analysis can be characterized with an aspect ratio distribution.

Inclusion Radius

The real size of the inclusions can be taken into account by Digimat through its radius which must be a strictly positive value. This value is only useful if a coating phase surrounding the inclusion phase is defined. The inclusion radius will also serve to compute the volume fraction or the mass fraction of the coating phase.

Orientation

The orientation definition is an important parameter of the inclusion phase characterization: it is the main reason for the anisotropy of the composite. It is represented by a unit vector p, aligned with the symmetry axis of the inclusion. There are four possibilities, described below, to define the orientation of the inclusions.

Fixed

All the inclusions contained in the RVE are aligned in the same direction. The orientation vector p is then defined by two spherical angles: θ and φ. θ is the angle in degrees between axis 3 and the orientation vector p, while φ is the angle in degrees between axis 1 and the projection of the orientation vector p onto the plane 1-2. Figure 7-8 illustrates p, θ and φ.

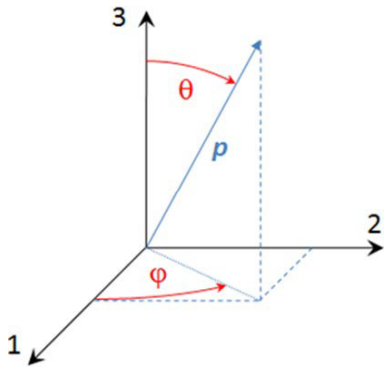


Figure 7-8 Axes definition based on the angles θ and φ, defining the orientation vector p.

In case of a composite reinforced with short fibers, the fibers often lie in the plane 1-2. In such case, θ should be close to 90° while φ could be any value between 0° and 180°, depending on the fibers main orientation in the plane 1-2.

Tensor: An orientation tensor is a simple but efficient way to describe an orientation distribution in the RVE. It is actually a simplification of what is called the orientation distribution function (ODF) $\varphi(p)$. The ODF gives the probability density to find an inclusion (in fact a spheroid) with the orientation vector p. For practical applications, the ODF contains too much information and it suffices to use the orientation tensor.

The orientation tensor a_{ij}, whose indices i and j go from 1 to 3, is defined as the second order moment of the ODF, namely

$$a_{ij} = \int p_i p_j \psi(p) dp \tag{7-7}$$

The orientation tensor is symmetric (the indices are permutable) such that only 6 components of the orientation tensor need to be defined. The diagonal terms indicate the intensity of the fiber orientation in the directions 1, 2 and 3. The non-diagonal terms are more complex to illustrate. They express the combination of a shift of the main peaks (i.e., intensities expressed by the diagonal terms) with a redistribution of the intensities in the angular space.

The redistribution of the intensities could be better understood by making the exercise of diagonalizing an orientation tensor in a global axis system to transfer it in the principal axis system. In Figure 7-9 are simple bi-dimensional examples showing that with the same initial values of the diagonal terms, it is possible to express both a random 2D orientation state and a fully aligned orientation in the 45° direction.

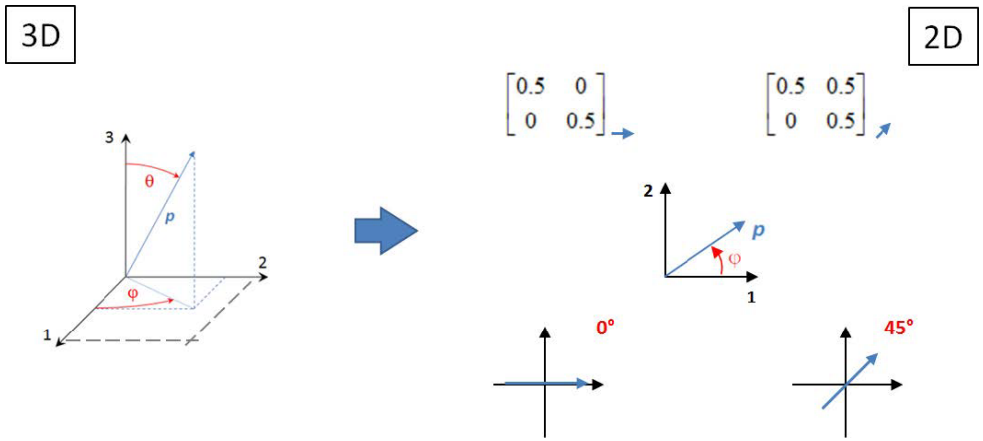


Figure 7-9 Random 2D distribution and a perfectly aligned state at 45°.

As Figure 7-9 shows on left side is illustrated a random 2D distribution while on right side is illustrated a perfectly aligned state at 45°. This is done keeping the same diagonal terms, varying only a_{12} from 0 to 0.5. The non-diagonal terms cannot be neglected since they influence more than could be imagined the anisotropy of the mechanical response of the composite. Notice the sign of the non-diagonal terms refers to the side of the shift in the axis system.

Remark: The orientation tensor must follow some rules that are described in details later on. Here is an overview of the main rules:

- The value of the diagonal terms a_{ii} must be in the range of 0 to 1.
- The sum of the diagonal terms, which is the trace invariant, must be equal to 1.
- The off-diagonal terms should have an absolute value lower than or equal to 0.5.
- If the off-diagonal terms are null, the default axis system is actually equivalent to the principal axis system (i.e., eigenvectors' directions of the diagonal tensor).
- If you want to define a fully aligned RVE (for instance a_{11}=1.0, a_{22}=a_{33}=0), it is advised to use the fixed orientation, instead of defining an orientation tensor, to reduce computation time.

Random 2D: The inclusions are randomly oriented in the plane 1-2. This is a particular case of an orientation tensor for which a_{11} and a_{22} are equal to 0.5 and all other terms are null.

Random 3D: The inclusions are randomly oriented in all three dimensions. This is a particular case of an orientation tensor for which all diagonal components are equal to 1/3 and all off-diagonal terms are null.

Remark: For a finite strain analysis involving elastic inclusions with a non-fixed orientation (orientation tensor, random 2D or random 3D), the orientation of the inclusions evolves during the loading. This is tracked by Digimat and the results are available in the inclusion result file.However, when hyperelastic inclusions with a non-fixed orientation definition are considered, their orientation will not evolve during the loading.

Orientation Tensor Statistics Rules

1. Trace invariant: $a_{11} + a_{22} + a_{33} = 1$

   If the sum is not equal to one but within a tolerance [1-tol; 1+tol], Digimat produces a warning and corrects the orientation tensor in the following way:

$$a = \begin{bmatrix} a_{11} & a_{12} & a_{13} \\ a_{12} & a_{22} & a_{23} \\ a_{13} & a_{23} & a_{33} \end{bmatrix} \Rightarrow a' = \begin{bmatrix} \frac{a_{11}}{\text{trace}(a)} & a_{12} & a_{13} \\ a_{12} & \frac{a_{22}}{\text{trace}(a)} & a_{23} \\ a_{13} & a_{23} & \frac{a_{33}}{\text{trace}(a)} \end{bmatrix} \tag{7-8}$$

   If the sum is outside of the defined tolerance, Digimat produces an error.

   When defining the orientation tensor directly in Digimat, the tolerance is set to 1E-3. The trace of the orientation tensor must then be in the range [0.999; 1.001].

When using orientation files, the user can define the tolerance, see the item tolerance on trace of orientation tensor under the **Integration Parameters** tab. The default tolerance value is 0.1.

2. The diagonal terms a_{ii} must be in the range of 0 to 1.

   All diagonal components of an orientation tensor must be between 0 and 1. If a component value is below 0 or above 1 but within the tolerance range (default or defined by the user), Digimat accepts it and sets it respectively to 0 or 1.

   If any of the diagonal components is not within the range of definition even considering the tolerance, Digimat produces an error and prints a message in its log file.

3. The off -diagonal terms a_{ij}, for $i \neq j$, must be in the range of -0.5 to 0.5.

   Off-diagonal components of the orientation tensor must have an absolute value within the range of 0 to 0.5. If the absolute value is above 0.5 but within the tolerance range (default or defined by the user), Digimat accepts it and sets it to 0.5. If one of the off-diagonal components is not within the range of definition even considering the tolerance, Digimat produces an error and prints a message in its log file.

For each of those orientation type, a graphical 3D representation is displayed. For fixed orientation, the direction of 1-axis the inclusions is displayed, while for the random and tensor orientation, the 3D ODF is displayed. For example, in case of a random 3D orientation, the ODF has the shape of a sphere.

Percolation Parameters

Percolation is a phenomenon that appears when having high volume fractions of inclusions. The inclusions actually start to touch each other inside the matrix phase and form chains, chains that can drastically increase the conductivity of a material. See Figure 7-10 for an illustration of the percolation phenomenon.

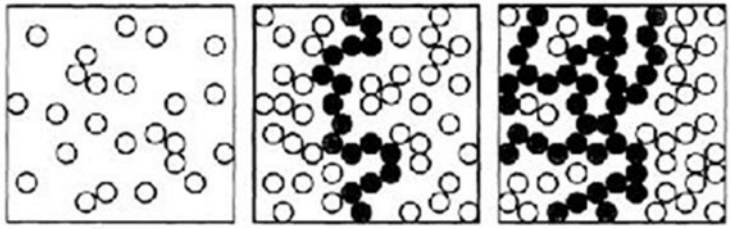


Figure 7-10 Evolution of the chains' formation, in a 2D percolating system.

Figure 7-10 shows evolution of the chains' formation, in a 2D percolating system, as the inclusion content increases, reaching the percolation threshold after which the conductivity highly increases Olivero and Radford (1998).

To capture the percolation effect in Digimat, the general percolation law has been implemented. This law introduces two parameters, namely the critical exponent t and the percolation threshold ϕ_c, i.e., the volume fraction at which percolation begins.

$$S_{comp} \approx S_{incl}\left(\frac{\phi - \phi_c}{1 - \phi_c}\right)^t \quad \text{for } \phi \geq \phi_c \tag{7-9}$$

The general percolation law is applicable to a wide range of mathematical and physical phenomena. It states that for an inclusion fraction ϕ that is greater than the critical fraction ϕ_c, also called the percolation threshold parameter, the composite property S_{comp} will approach the inclusion property S_{incl} according to the factor in brackets, which goes from 0 to 1 for an increasing phase fraction ϕ.

The percolation parameters are available only for thermal and electrical types of analyses. Percolation can be applied, in general, to any analysis involving a material law described by second order tensors, such as the thermal conductivity tensor (Fourier Law) and the electrical conductivity tensor (Ohm Law). In experimental practice, the percolation phenomenon is observed mainly for electrical conduction.

Percolation threshold: This parameter ϕ_c refers to the critical fraction of inclusions at which percolation begins. The percolation threshold can be expressed either as a volume fraction or as a mass fraction; it must be expressed in the same system as that of the phase fraction. The percolation model is active only for inclusion fractions that are greater than the percolation threshold. Thus, if the inclusion fraction is smaller than the percolation threshold, the percolation model will be turned off automatically (see the message printed in the `*.log` file).

A value for the percolation threshold is obtained from a fitting procedure using experimental data, or from Monte Carlo simulations. For spherical inclusions, the percolation threshold will generally have a value of around 0.30, while for high aspect ratios, the percolation threshold can go down to very low values like 0.01 or even smaller.

Percolation exponent: The percolation exponent refers to the critical exponent t in the general percolation law and it determines if the factor in brackets goes from 0 to 1 quickly or slowly. A value for the percolation exponent is also obtained from a fitting procedure using experimental data, or from Monte Carlo simulations.

For 3D continuum percolation, the exponent t is usually between 2 and 3. For 2D percolation, typical values of the exponent t are lower than for 3D percolation.

Coating

Coatings are accessible for the elastic inclusions and for voids. It can be used when having an elastic or a viscoelastic matrix, and it can be assigned either an elastic or a viscoelastic material. The **Coating** tab with the available coating parameters are shown in the Figure 7-11.

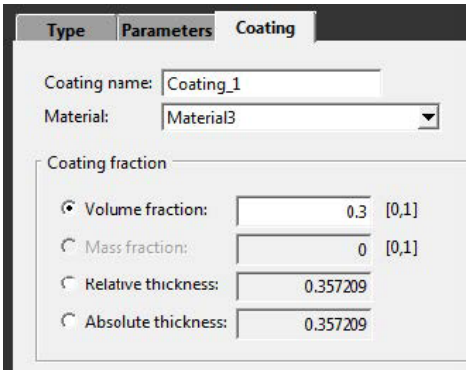


Figure 7-11 Coating tab with the available coating parameters.

Notice that to each new coating must be assigned a unique name, a material and a coating fraction. The latter can be defined by a volume fraction, a mass fraction, a relative thickness (relative to the inclusion size) or an absolute thickness.

Specifying a mass fraction is only possible if the material density has been defined and if the inclusion phase fraction is also defined in mass. If selecting the relative thickness, you must know it actually refers to the coating thickness divided by the inclusion's radius. Finally, if you want to define the coating with an absolute thickness, you first have to define the absolute inclusion's radius. The radius is the half length of one of the two minor axis of the ellipsoidal inclusion

Remarks:

- The shape and the orientation definition are not given for the coating since it receives the same definition as that of the inclusion it surrounds.
- When coatings are considered, the homogenization multi-level method can overcome some deficiencies of the default multi-step method. Therefore the multi-level method is advised for such case.
- Only one coating can surround an inclusion phase, which means that it is not yet possible to model multi-skins particles having different materials on each of these skins.

Clustering

Clustering is accessible only for elastic inclusions. It can be used when having an elastic or an elasto-plastic matrix. The **Clustering** tab is described in Figure 7-12.

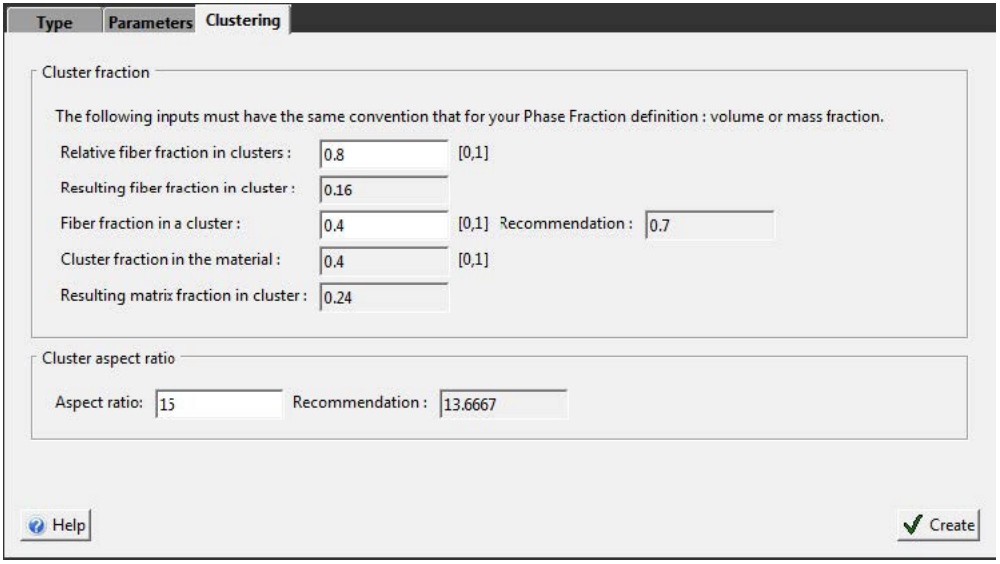


Figure 7-12 Clustering tab with the available clustering parameters.

Notice that only one cluster can be assigned to a phase. The latter can be defined by a relative fiber fraction in cluster in the RVE, a fiber fraction in a cluster and an aspect ratio. The cluster orientation is assumed equivalent to the one of the inclusions.

Once a relative fiber fraction in clusters is chosen, recommended values of the fiber fraction in a cluster and the aspect ratio are computed. These recommended values are based on observations made when generating RVE of long fiber materials in Digimat-FE. Three additional information are shown:

- Resulting fiber fraction in cluster
- Cluster fraction in the material
- Resulting matrix fraction in cluster

They are useful to improve understanding the material generated.

Remarks:

- The phase and his cluster will be transformed into two distinct phases in the solver.
- When clustering is considered, the homogenization multi-level method can overcome some deficiencies of the default multi-step method. Therefore the multi-level method is advised for such case.

- A number of angle increments of 32 is advised, particularly when the final goal is to use the Hybrid method on a structural applications.
- The reverse engineering of cluster parameters has to be done in Digimat-MF in order to correctly predict the elastic properties of your composite. The plastic parameters of your matrix and the failure parameters can then be reverse engineered as usual in Digimat-MX.
- Coatings and clustering cannot be combined.
- Clustering cannot be used in multi-layer analysis.

Yarn Phase Parameters (Advanced Yarn)

A yarn phase is similar to a continuous fiber phase, in the sense that aspect ratio is considered to be infinite.

However, since a yarn is actually made from a set of filament (also called fibers), some extra information is necessary. This extra information is related to the way the different fibers are set together to form a yarn:

- Yarn density: can be expressed as the density in tex (gram per kilometer) or as the number of fibers in one yarn
- Fiber data: the diameter of one individual fiber (assumed to be of circular cross-section)
- Yarn cross-section: an elliptical cross-section is assumed for the yarn. Therefore, two diameters have to be specified

Based on these three inputs, it is possible to compute the fraction of fibers inside a yarn.

The material assigned to this type of phase should be understood as the material of the filaments composing the yarn, not of the complete yarn. Indeed, once processed, the yarn will actually be a composite in itself, with matrix material.

8 RVE

- RVE Types
- Single Microstructure
- Multilayer Microstructure

RVE Types

This section introduces the RVE types that can be defined in Digimat-MF. The different options for each type of RVE are presented.

Two types of RVE are available:

- Single microstructure
- Multilayer microstructure

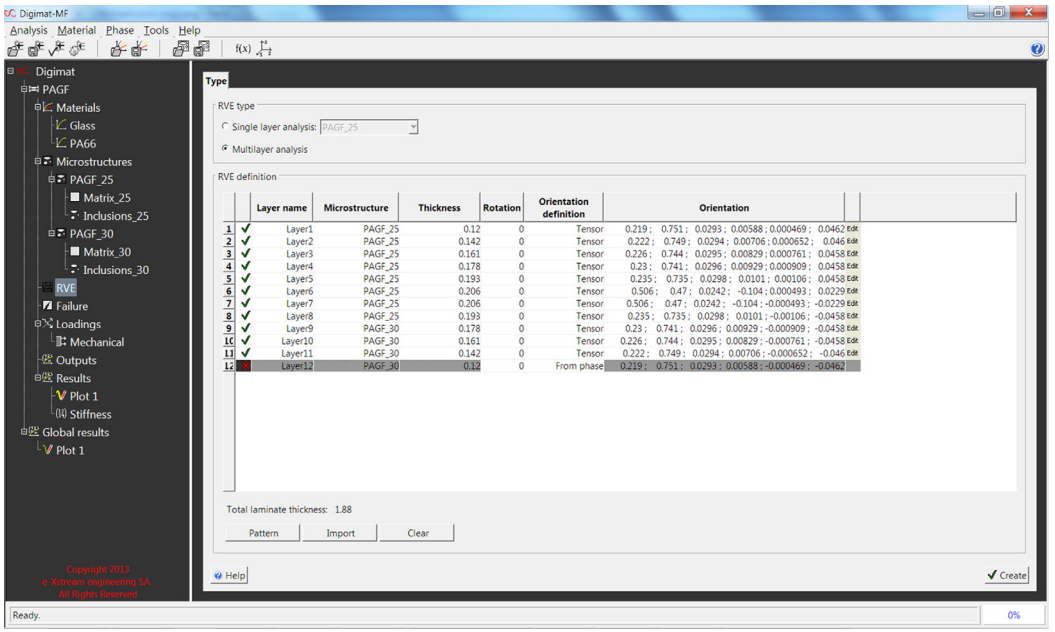


Figure 8-1 RVE definition interface window.

Single Microstructure

Two different kinds of microstructures can be created in Digimat-MF. The first one corresponds to a **Single microstructure analysis**.

In this case, the RVE is made up of a matrix phase reinforced with one or several inclusion phases. The user must associate the name of a microstructure, defined under the **Microstructures** tree item, with the RVE.

By default, the first microstructure item, in alphabetical order of all the created microstructures, is associated with the RVE.

Multilayer Microstructure

Two different kinds of microstructures can be created in Digimat-MF. The second one corresponds to a **Multilayer microstructure analysis**.

In this case, the RVE consists of a stack of layers. With each layer is associated a previously defined microstructure. Moreover, in the RVE definition, the inclusions' orientations can be redefined for each layer without modifying the microstructure parameters.

Several options for creating a multilayer RVE are available. Their description is provided in the following sections. For an overview of the graphical interface, see Figure 8-1.

Layer Attributes

Each layer of the multilayer RVE is defined by the following attributes:

- **Layer name**: this text field defines the name of the layer. By default, LayerN is the name of the N^{th} layer. As layers are identified by their names, each layer must have a different name.
- **Microstructure**: this drop-down menu gives the user the possibility to choose among the already defined microstructures the one that will serve as definition for the current layer. By default, the orientation of the layer is taken as the one being defined in the inclusion phase of the microstructure. There are however two ways to modify the orientation of the inclusion phase without creating another microstructure:
  - **Rotation**: this text field can be used to define an additional rotation, in the plane 1-2, that will be applied to the inclusion phase orientation definition. This only applies to orientations specified as fixed or by a tensor.
  - **Orientation definition**: five types of orientation can be attributed to the layer, overriding or not the orientation definition that is given in the microstructure definition.
    - **From phase**: the orientation of the inclusion phase is given by the one defined in the microstructure associated with the current layer.
    - **Fixed**: allows the user to define the spherical angles φ and θ that describe the orientation of the fibers by opening the dialog box in the **Orientation** column. This option overrides the orientation defined in the microstructure item.
    - **Random 2D/Random 3D**: these choices can be made to assign a random distribution of the fiber orientation, either in the plane 1-2 or in the RVE 3D space. This option overrides the orientation defined in the microstructure item.

- **Tensor**: allows the user to define the orientation tensor that describes the orientation of the fibers by opening the dialog box in the **Orientation** column. This option overrides the orientation defined in the microstructure item.

The same microstructure can be attributed to different layers.

- **Thickness**: this text field should be used to define the real thickness of the layer or relative to the overall RVE thickness. This value must be strictly positive. No pre-defined values are used.
- **Orientation**: this field reports the spherical angles if the orientation is said to be fixed, or it reports the orientation tensor components entered by the user when redefining the inclusion phase's orientation.
  Note that an additional rotation can be applied to the fixed orientation or the orientation tensor defined in this field. To edit this field, one must click **Edit** button located at the end each layer's row.

Note that the sole layers with the green check symbol will be taken into account in the computation. To activate or deactivate a layer, click on the symbol in the leftmost column of the RVE definition table. A deactivated layer is grayed and marked with a red cross in the leftmost column.

Multilayer RVE Creation: Methodology

Three complementary methods are available to create a laminate material in Digimat-MF GUI. These methods are described here after.

Method 1: Right-clicking

By right-clicking in the RVE definition table while hovering over a layer, called the selected layer in the following, the **Layer Context Menu** appears. This menu gives the user several options to add, delete, copy or move layers in the RVE definition table.

All these operations are performed with respect to the selected layer.

Method 2: Using the Pattern Tool

Left-clicking the **Pattern** button at the bottom of the RVE definition table (see Figure 8-1), opens a dialog box enabling the user to create a multilayer microstructure using copy/symmetry/anti-symmetry tools.

- The **Copy** pattern tool: after having selected the layers of the microstructure to be copied, the user can choose the number of times the pattern should be copied and where they should be copied in the already existing structure.

- The **Anti-symmetry** and **Symmetry** pattern tools: using these two options, the user can construct an anti-symmetric or symmetric laminate. After selecting the layers to be patterned, the type of symmetry and the layer of symmetry, the multilayer microstructure is updated by clicking the **OK** button.

Method 3: Import from Orientation File

Left-clicking on the **Import** button located below the RVE definition table opens a dialog box offering the user to load a multilayer RVE definition from the formatted files (see Figure 8-2).

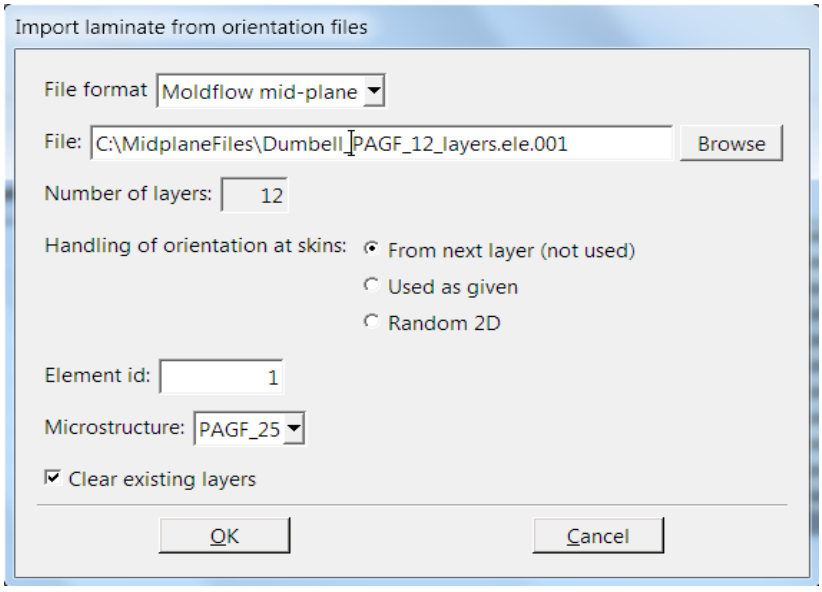


Figure 8-2 Import of a multilayer RVE definition from midplane files.

Several options are available for the import:

- **Available file formats**:
  - Moldflow Midplane (`*.xml`, `*.ele`)
  - Moldex3D Midplane (`*.ele`)
  - 3D TIMON Midplane (`*.dat`)
  - Digimat (`*.dof`)
  - CSV file (`*.csv` file) (see below for description and specificities of this format)
- **File**: this field allows the user to specify the path to the file(s) containing the RVE definition.

- **Handling of orientation at skins** (only for Moldflow data): the orientation tensors coming from an orientation file are given at each layer skin and not for each layer, i.e., there are N+1 skins for an N layer RVE. Except for external layers, the orientation tensor of a layer is computed by averaging the orientation tensors given at the layer skins. For external layers, three options are available to compute the orientation tensor:
  - **From next layer (not used)**: using this option will attribute the orientation tensors of skins 2 and N to layers 1 and N. This is the default method.
  - **Used as given**: using this option will compute the orientation tensors of the external layers by averaging the orientation tensors from their skins.
  - **Random 2D**: using this option will compute the orientation in external layers using a Random 2D orientation definition in the extreme skins.
- **Element id**: this allows the user to select which orientation tensors should be extracted. The default value is 1. For Digimat orientation files (`*.dof`) which can contain 2^{nd} order elements' orientation definition, the first integration point of the chosen element is selected.
- **Microstructure**: name of the microstructure to be associated to the imported laminate.

Note that the number of layers and the thickness distribution employed in the Moldflow Midplane simulation are automatically identified by the import module. If a laminate is already defined, these can be erased at the **Clear existing layers** box is checked.

CSV Laminate File (*.csv) is a specific orientation file format that contains the definition of a single laminate (or multilayer RVE). Using this format deactivates the **Handling of orientation at skins** and **Element Id** frameboxes. The CSV Laminate File format convention is the following:

```
LayerName, Thickness, Rotation,a11,a22,a33,a12,a13,a23
LayerName, Thickness, Rotation, theta, phi
# Hash - starting and empty lines are ignored.
# Each line must contain 5 or 9 entries.
# The entries can be separated by semicolons, commas, blankspaces or
tabulations.
# All entries are numeric values, except LayerName (text without
blankspaces).
# Thickness must be strictly positive.
# AdditionalRotation, theta and phi are expressed in degrees.
```

Using this file format enables the user to define each layer's name, thickness, rotation and orientation. The orientation definition is automatically switched between "Fixed" and "Tensor" when doing so; however, other orientation definitions are not managed.

9 Failure

- Failure Indicators
- Failure Criterion Definition & Assignment
- Progressive Failure Model
- First Pseudo-Grain Failure Model
- Fatigue Models
- Matrix Damage Fatigue Model

Failure Indicators

This section introduces the failure indicators that are implemented in Digimat. Failure indicators are real-valued functions comparing a given stress (strain) state combination to strengths (failure criteria). They are written in a normalized dimensionless form in such a way that an indicator value smaller than 1 means a safe state, while failure is deemed to occur as soon as the chosen indicator reaches or exceeds 1. Failure indicators are used as a post-processing tool to identify critical or failing zones in a Digimat-CAE analysis or failure of material point in a Digimat-MF analysis.

Remark: Unless you use the First Pseudo-Grain Failure scheme (FPGF) (see section First Pseudo-Grain Failure Model), the failure indicators do not influence the material response.

The failure criteria presented in this section were proposed in the literature for laminated composites (mainly used in aerospace) in order to predict failure at the ply level. Therefore, they have been historically used for composites such that as a thermoset polymer matrix (e.g., epoxy is reinforced with continuous and aligned stiff fibers (e.g., carbon fibers). Those indicators are used in a post-processing manner at each ply level.

In other words, the stress state in the ply is computed without any coupling with any damage mechanism within the ply. Those failure indicators are consequently valid when failure of the ply can be considered as being essentially of brittle nature, with negligible plastic deformation at the ply level. In Digimat-MF, failure indicators are available for general composite systems, including ones with significant ductile matrix behavior (e.g., thermoplastic polymer matrix). The indicators have been generalized in Digimat-MF to composite which are not laminates (no plies) and which are reinforced with misaligned short fibers. This generalization was carried out thanks to the First Pseudo-Grain Failure (FPGF) method (see section First Pseudo-Grain Failure Model). However, the limitations outlined above should be kept in mind and the criteria of this section are to be considered as simple although useful indicators.

As of today, following are the available failure indicators in Digimat:

- Maximum component (stress-based or strain-based)
- Tsai-Hill 2D (stress-based or strain-based)
- Tsai-Hill 3D transversely isotropic (stress-based or strain-based)
- Tsai-Hill 3D (stress-based or strain-based)
- Azzi-Tsai-Hill 2D
- Tsai-Wu 2D (stress-based or strain-based)
- Tsai-Wu 3D transversely isotropic (stress-based or strain-based)
- Tsai-Wu 3D orthotropic
- Tsai-Wu 3D
- Multi-component 2D
- Multi-component 3D Orthotropic
- Hashin-Rotem 2D

- Hashin 2D
- Hashin 3D
- SIFT
- Christensen
- Camanho
- Accumulated plastic strain
- User-defined

Unless notified otherwise, criteria are only stress based.

Each of this models requires its own set of strength parameters to be defined (even though some of them are common to several models). Some strength parameters can be estimated based on Kelly-Tyson's formula.

For any failure indicator, it is possible to define dependencies over the strength parameters (see Setting Dependencies), e.g., a to make them dependent of the strain rate, the temperature, or of some User-defined variables.

Note also that all failure indicators can be applied at the pseudo-grain level, using the FPGF scheme, in any axis system. In the following and unless otherwise indicated though, the strain and stress components as well as the strength parameters are defined in a local axis system such that direction 1 corresponds to the fibers axis, direction 2 is perpendicular to the fibers axis and is in the "ply plane", and direction 3 is orthogonal to that plane (see Figure 9-1). However, in the case of 2D local FPGF criteria, this default definition can be overridden by the **Use critical basis** option.

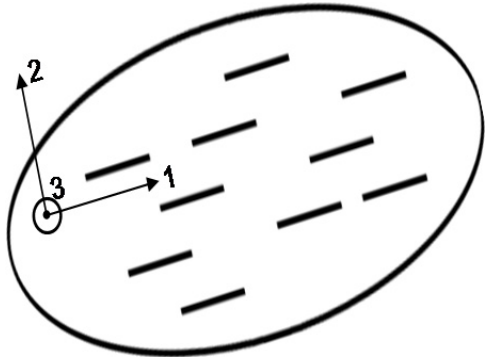


Figure 9-1 Local axis system definition generally used when working with the FPGF scheme.

General Definition

Failure criteria are defined by one or several strength parameters; for instance, the Tsai-Wu failure criteria are defined by the following strength parameters:

1. $X_t \rightarrow X_1^t$: tensile strength in the 1-direction
2. $X_c \rightarrow X_1^c$: compressive strength in the 1-direction
3. $Y_t \rightarrow Y_2^t$: tensile strength in the 2-direction
4. $Y_c \rightarrow Y_2^c$: compressive strength in the 2-direction
5. $S \rightarrow S_6$: shear strength in the plane 1-2

Remark: Some of the most classical failure indicators were initially designed for usage at a thin ply level, and they assume a plane stress state where the only non-zero ply-level stresses are σ_{11}, σ_{22} and σ_{33}.

Most of the failure criteria defined in Digimat are based on stress tensors, which provides an intuitive interpretation of their strength parameters from experimental stress-strain curves in typical directions. Some of these criteria were also given strain-based counterparts having a similar expression, but using the strain components instead of the stress components.

Remark: When defining a strain-based failure indicator with interactive strain components (typically, Tsai- Hill and Tsai-Wu models), the user shall keep in mind that the maximum strain parameters do **not** correspond to the maximum strains obtained from a conventional tensile test, due to the transverse shrinkage effects (related to the Poisson's ratio).

In Digimat, the failure surface is described mathematically by a function defined in the stress-strain space. More precisely, failure is deemed to occur when the so-called failure function reaches 1, i.e., $F(\sigma, \varepsilon) \geq 1$. At each time step, Digimat outputs the value of a failure indicator f whose value is defined implicitly as the positive solution of the (non)linear equation

$$F\left(\frac{\sigma}{f}, \frac{\varepsilon}{f}\right) = 1$$

Hence, the failure indicator f can be interpreted as the inverse of a security factor; its definition implies that $f = 1$ when the current mechanical state is on the failure surface, i.e., $F(\sigma, \varepsilon) = 1$. In the following particular examples, the failure indicator f can be computed explicitly:

- $f = F(\sigma)$ for stress or strain-component failure indicators, e.g., $F(\sigma) = \max\left\{\frac{\sigma_{11}}{X_t}, 0\right\}$
- $f = \sqrt{F(\sigma)}$ for Tsai-Hill failure indicators that can be written as $F(\sigma) = \sigma : F : \sigma$
- f is the positive root of the second-order equation $\frac{\sigma : F : \sigma}{f^2} + \frac{C : \sigma}{f}$ for Tsai-Wu failure indicators written as $F(\sigma) = \sigma : F : \sigma + c : \sigma$.

Remark: Before version 5.0.1 of Digimat, the failure indicators could have linear, quadratic, or even mixedmode shapes, which made quantitative interpretation sometimes difficult. The linear formulation overcomes this issue and leads to linear evolutions of the failure indicators with stress

and strain levels, whichever the shape of the failure functions. This linear formulation is activated by default; however, the old formulation is still available through the Digimat-MF GUI, by unchecking the **Use linear formulation** checkbox in the **Failure Indicator Definition** tab. The results will typically differ if the failure indicator critical value if different from one, or if "mixed-mode" failure criteria are used, or if non-linear-shaped failure indicators are used with the FPGF scheme.

Stress Component

With the stress component failure indicator, there is no interaction between different stress components, each one is considered separately. More sophisticated indicators are presented elsewhere in this section. This criterion has three input arguments and outputs two indicators.

Input:

1. The component of the stress tensor used in the indicator, e.g., the 11-component.
2. The tensile strength, $X_t > 0$.
3. The compressive strength, $X_c > 0$.

The strengths have the dimension of a stress.

Output:

1. Tensile failure indicator:

$$f_A = F_A(\sigma) \text{ with } F_A(\sigma) = \frac{\sigma_{ij}}{X_t} \text{ if } \sigma_{ij} > 0, \text{ 0 otherwise} \tag{9-1}$$

2. Compressive failure indicator:

$$f_B = F_B(\sigma) \text{ with } F_B(\sigma) = -\frac{\sigma_{ij}}{X_c} \text{ if } \sigma_{ij} < 0, \text{ 0 otherwise} \tag{9-2}$$

If the failure indicator is computed in the principal axes of the stress tensor, only the diagonal terms of the tensor are non-zero. In that case, the eigenvalues are ordered as follows:

$$\sigma_{11} \geq \sigma_{22} \geq \sigma_{33} \tag{9-3}$$

The outputs of the failure indicator then become:

$$f_A = F_A(\sigma) \text{ with } F_A(\sigma) = \frac{\sigma_1}{X_t} \text{ if } \sigma_1 > 0, \text{ 0 otherwise} \tag{9-4}$$

$$f_B = F_B(\sigma) \text{ with } F_B(\sigma) = -\frac{\sigma_3}{X_c} \text{ if } \sigma_3 < 0, \text{ 0 otherwise} \tag{9-5}$$

Failure happens if (at least) one of these two indicators reaches or exceeds the critical value of 1.

Strain Component

With the strain component failure indicator, there is no interaction between different strain components, each one is considered separately. More sophisticated indicators are presented elsewhere in this section. This criterion has three input arguments and outputs two indicators (as for the stress component failure criterion).

Input:

1. The component of the strain tensor used in the indicator, e.g., the 11-component.
2. The maximum tensile strain, $X_t > 0$.
3. The maximum compressive strain, $X_c > 0$.

The maximum strains have no dimension.

The user shall keep in mind that these parameters **do not always** correspond to the maximum strains obtained from a conventional tensile test, due to the transverse shrinkage effects (related to the Poisson's ratio). This is especially the case when using the criterion in principal axis system.

Output:

1. Tensile failure indicator:

$$f_A = F_A(\varepsilon) \text{ with } F_A(\varepsilon) = \frac{\varepsilon_{ij}}{X_t} \text{ if } \varepsilon_{ij} > 0, \text{ 0 otherwise} \tag{9-6}$$

2. Compressive failure indicator:

$$f_B = F_B(\varepsilon) \text{ with } F_B(\varepsilon) = -\frac{\varepsilon_{ij}}{X_c} \text{ if } \varepsilon_{ij} < 0, \text{ 0 otherwise} \tag{9-7}$$

If the failure indicator is computed in the principal axes of the strain tensor, only the diagonal terms of the tensor are non-zero. In that case, the eigenvalues are ordered as follows:

$$\varepsilon_{11} \geq \varepsilon_{22} \geq \varepsilon_{33} \tag{9-8}$$

The outputs of the failure indicator then become:

$$f_A = F_A(\varepsilon) \text{ with } F_A(\varepsilon) = \frac{\varepsilon_1}{X_t} \text{ if } \varepsilon_1 > 0, \text{ 0 otherwise} \tag{9-9}$$

$$f_B = F_B(\varepsilon) \text{ with } F_B(\varepsilon) = -\frac{\varepsilon_3}{X_c} \text{ if } \varepsilon_3 < 0, \text{ 0 otherwise} \tag{9-10}$$

Failure happens if (at least) one of these two indicators reaches or exceeds the critical value of 1.

Tsai-Hill 2D

This criterion has five input arguments and outputs one indicator.

Input:

1. Axial tensile strength, $X_t > 0$
2. Axial compressive strength, $X_c > 0$
3. In-plane tensile strength, $Y_t > 0$
4. In-plane compressive strength, $Y_c > 0$
5. Transverse shear strength, $S > 0$.

The strengths have the dimension of a stress.

Output:

1. Failure indicator:

$$f_A = \sqrt{F_A(\sigma)} \text{ with } F_A(\sigma) = \frac{\sigma_{11}^2}{X^2} - \frac{\sigma_{11}\sigma_{22}}{X^2} + \frac{\sigma_{22}^2}{Y^2} + \frac{\sigma_{12}^2}{S^2} \tag{9-11}$$

the strengths being taken in tension or in compression depending on the sign of their respective component ($\sigma_{11} \leftrightarrow X$ and $\sigma_{22} \leftrightarrow Y$). This is a limiting assumption of the criterion. Another simplification is to consider only the longitudinal strength X in the denominator of the product $\sigma_{11} \times \sigma_{22}$. Failure is deemed to occur when the indicator reaches or exceeds the value of 1.

Tsai-Hill 3D Transversely Isotropic

This criterion assumes a material isotropy in the plane normal to axis 1 (default), 2 or 3. It has three input arguments (as it assumes identical strengths in tension and compression) and outputs one indicator.

Input:

1. Axial tensile strength, $X > 0$
2. In-plane tensile strength, $Y > 0$
3. Transverse shear strength, $S > 0$.

The strengths have the dimension of a stress.

Output:

1. Failure indicator:

$$f_A = \sqrt{F_A(\sigma)} \text{ with}$$

$$F_A(\sigma) = \frac{\sigma_{11}^2}{X^2} - \frac{\sigma_{11}(\sigma_{22}+\sigma_{33})}{X^2} + \frac{\sigma_{22}^2+\sigma_{33}^2}{Y^2} + \left(\frac{1}{X^2} - \frac{2}{Y^2}\right)\sigma_{22}\sigma_{33} + \frac{\sigma_{12}^2+\sigma_{13}^2}{S^2} + \left(\frac{4}{Y^2} - \frac{1}{X^2}\right)\sigma_{23}^2$$

when the normal to the plane of isotropy corresponds to axis 1

$$F_A(\sigma) = \frac{\sigma_{22}^2}{X^2} - \frac{\sigma_{22}(\sigma_{11}+\sigma_{33})}{X^2} + \frac{\sigma_{11}^2+\sigma_{33}^2}{Y^2} + \left(\frac{1}{X^2} - \frac{2}{Y^2}\right)\sigma_{11}\sigma_{33} + \frac{\sigma_{12}^2+\sigma_{23}^2}{S^2} + \left(\frac{4}{Y^2} - \frac{1}{X^2}\right)\sigma_{13}^2$$

when the normal to the plane of isotropy corresponds to axis 2

$$F_A(\sigma) = \frac{\sigma_{33}^2}{X^2} - \frac{\sigma_{33}(\sigma_{11}+\sigma_{22})}{X^2} + \frac{\sigma_{11}^2+\sigma_{22}^2}{Y^2} + \left(\frac{1}{X^2} - \frac{2}{Y^2}\right)\sigma_{11}\sigma_{22} + \frac{\sigma_{13}^2+\sigma_{23}^2}{S^2} + \left(\frac{4}{Y^2} - \frac{1}{X^2}\right)\sigma_{12}^2 \quad (9\text{-}12)$$

when the normal to the plane of isotropy corresponds to axis 3

Failure is deemed to occur when the indicator reaches or exceeds the value of 1.

Tsai-Hill 3D

This criterion generalizes Tsai-Hill 2D from plane stress to 3D stress state. It has six input parameters and outputs one indicator.

Input:

1. Axial tensile strength (in direction 1), $X > 0$
2. In-plane tensile strength (in direction 2), $Y > 0$
3. Out-of-plane tensile strength (in direction 3), $Z > 0$
4. Shear 12 strength, $S_{xy} > 0$
5. Shear 23 strength, $S_{yz} > 0$
6. Shear 13 strength, $S_{xz} > 0$

These strengths have the dimension of a stress.

To ensure full closure of the failure surface, the following relations must be satisfied:

$$F_{XY} = -0.5 \cdot \left(\frac{X}{Y} + \frac{Y}{X} - \frac{X \cdot Y}{Z \cdot Z}\right) < 1 \quad (9\text{-}13)$$

$$F_{YZ} = -0.5 \cdot \left(\frac{Z}{Y} + \frac{Y}{Z} - \frac{Y \cdot Z}{X \cdot X}\right) < 1 \quad (9\text{-}14)$$

$$F_{XZ} = -0.5 \cdot \left(\frac{X}{Z} + \frac{Z}{X} - \frac{X \cdot Z}{Y \cdot Y}\right) < 1 \tag{9-15}$$

Output:

1. Failure indicator:

$$f_A = \sqrt{F_A(\sigma)} \quad \text{with}$$

$$F_A(\sigma) = \frac{\sigma_{11}^2}{X^2} + \frac{\sigma_{22}^2}{Y^2} + \frac{\sigma_{33}^2}{Z^2} + \frac{\sigma_{12}^2}{S_{xy}^2} + \frac{\sigma_{23}^2}{S_{yz}^2} + \frac{\sigma_{13}^2}{S_{xz}^2} - \left(\frac{1}{X^2} + \frac{1}{Y^2} - \frac{1}{Z^2}\right)\sigma_{11}\sigma_{22} - \left(-\frac{1}{X^2} + \frac{1}{Y^2} + \frac{1}{Z^2}\right)\sigma_{22}\sigma_{33} - \left(\frac{1}{X^2} - \frac{1}{Y^2} + \frac{1}{Z^2}\right)\sigma_{11}\sigma_{33} \tag{9-16}$$

Remark:

Prior to Digimat 2021.4, the failure indicator was expressed by the equivalent expression:

$$f_A = \sqrt{F_A(\sigma)} \quad \text{with}$$

$$F_A(\sigma) = A(\sigma_{11} - \sigma_{22})^2 + B(\sigma_{22} - \sigma_{33})^2 + C(\sigma_{33} - \sigma_{11})^2 + 2L\sigma_{12}^2 + 2M\sigma_{23}^2 + 2N\sigma_{13}^2 \tag{9-17}$$

with the strength parameters A, B, C, L, M, and N such that:

$A + B > 0$, $A + C > 0$, $B + C > 0$

$L > 0$, $M > 0$, $N > 0$

The relations between the two sets of parameters are the following:

$$A = \frac{1}{2}\left(\frac{1}{X^2} + \frac{1}{Y^2} - \frac{1}{Z^2}\right), \quad B = \frac{1}{2}\left(-\frac{1}{X^2} + \frac{1}{Y^2} + \frac{1}{Z^2}\right), \quad C = \frac{1}{2}\left(\frac{1}{X^2} - \frac{1}{Y^2} + \frac{1}{Z^2}\right)$$

$$L = \frac{1}{2S_{xy}^2}, \quad M = \frac{1}{2S_{yz}^2}, \quad N = \frac{1}{2S_{xz}^2}$$

OR

$$X = \frac{1}{\sqrt{A + C}}, \quad Y = \frac{1}{\sqrt{A + B}}, \quad Z = \frac{1}{\sqrt{B + C}}$$

$$S_{xy} = \frac{1}{\sqrt{2L}}, \quad S_{yz} = \frac{1}{\sqrt{2M}}, \quad S_{xz} = \frac{1}{\sqrt{2N}}$$

When loading an analysis file that contains a failure indicator with this alternative input format, the strength parameters A-B-C-L-M-N are automatically converted to X-Y-Z-S_{ij} strengths, if no

dependencies are attached to those parameters. Otherwise, a dedicated button enables to make the conversion, but also discards the dependencies.

Tsai-Hill 2D, 3D Transversely Isotropic and 3D (Strain-based)

The three former Tsai-Hill criteria are based on the components of the stress tensor, with different formulations depending on the required material symmetry. On the other hand, one can define similar failure indicators based on the components of the strain tensor, which is also meaningful from a material point of view. Thus, three more criteria are available for using in Digimat-MF:

- Tsai-Hill 2D Strain-based

  This failure indicator is the counterpart of the previous Tsai-Hill 2D criterion. Its expression is:

$$f_A = \sqrt{F_A(\varepsilon)} \text{ with } F_A(\varepsilon) = \frac{\varepsilon_{11}^2}{X^2} - \frac{\varepsilon_{11}\varepsilon_{22}}{X^2} + \frac{\varepsilon_{22}^2}{Y^2} + \frac{(2\varepsilon_{12})^2}{S^2} \tag{9-18}$$

  where X, Y and S are unitless maximum strains parameters (instead of strength).

- Tsai-Hill 3D Transversely Isotropic Strain-based

  This failure indicator is the counterpart of the previous Tsai-Hill 3D Transversely Isotropic criterion. Its expression is:

$$f_A = \sqrt{F_A(\varepsilon)} \text{ with}$$

$$F_A(\varepsilon) = \frac{\varepsilon_{11}^2}{X^2} - \frac{\varepsilon_{11}(\varepsilon_{22}+\varepsilon_{33})}{X^2} + \frac{\varepsilon_{22}^2+\varepsilon_{33}^2}{Y^2} + \left(\frac{1}{X^2} - \frac{2}{Y^2}\right)\varepsilon_{22}\varepsilon_{33} + \frac{(2\varepsilon_{12})^2+(2\varepsilon_{13})^2}{S^2} + \left(\frac{1}{Y^2} - \frac{1}{4X^2}\right)(2\varepsilon_{23})^2$$

  when the normal to the plane of isotropy corresponds to axis 1

$$f_A = \sqrt{F_A(\varepsilon)} \text{ with}$$

$$F_A(\varepsilon) = \frac{\varepsilon_{22}^2}{X^2} - \frac{\varepsilon_{22}(\varepsilon_{11}+\varepsilon_{33})}{X^2} + \frac{\varepsilon_{11}^2+\varepsilon_{33}^2}{Y^2} + \left(\frac{1}{X^2} - \frac{2}{Y^2}\right)\varepsilon_{11}\varepsilon_{33} + \frac{\varepsilon_{12}^2+\varepsilon_{23}^2}{S^2} + \left(\frac{4}{Y^2} - \frac{1}{X^2}\right)\varepsilon_{13}^2$$

  when the normal to the plane of isotropy corresponds to axis 2

$$f_A = \sqrt{F_A(\varepsilon)} \text{ with}$$

$$F_A(\varepsilon) = \frac{\varepsilon_{33}^2}{X^2} - \frac{\varepsilon_{33}(\varepsilon_{11}+\varepsilon_{22})}{X^2} + \frac{\varepsilon_{11}^2+\varepsilon_{22}^2}{Y^2} + \left(\frac{1}{X^2} - \frac{2}{Y^2}\right)\varepsilon_{11}\varepsilon_{22} + \frac{\varepsilon_{13}^2+\varepsilon_{23}^2}{S^2} + \left(\frac{4}{Y^2} - \frac{1}{X^2}\right)\varepsilon_{12}^2 \tag{9-19}$$

  when the normal to the plane of isotropy corresponds to axis 3

  where X, Y and S are unitless maximum strains parameters (instead of strengths parameters).

- Tsai-Hill 3D Strain-based

This failure indicator is the counterpart of the previous Tsai-Hill 3D criterion. Its expression is:

$$f_A = \sqrt{F_A(\varepsilon)} \qquad \text{with}$$

$$F_A(\varepsilon) = \frac{\varepsilon_{11}^2}{X^2} + \frac{\varepsilon_{22}^2}{Y^2} + \frac{\varepsilon_{33}^2}{Z^2} + \frac{(2\varepsilon_{12})^2}{S_{xy}^2} + \frac{(2\varepsilon_{23})^2}{S_{yz}^2} + \frac{(2\varepsilon_{13})^2}{S_{xz}^2} - \left(\frac{1}{X^2} + \frac{1}{Y^2} - \frac{1}{Z^2}\right)\varepsilon_{11}\varepsilon_{22} - \left(-\frac{1}{X^2} + \frac{1}{Y^2} + \frac{1}{Z^2}\right)\varepsilon_{22}\varepsilon_{33} - \left(\frac{1}{X^2} - \frac{1}{Y^2} + \frac{1}{Z^2}\right)\varepsilon_{11}\varepsilon_{33} \tag{9-20}$$

where X, Y, Z, S_{xy}, S_{yz}, S_{xz} are unitless maximum strains parameters (instead of strengths).

All these criteria have the same number of inputs and outputs as their stress-based counterparts. Moreover, the user shall keep in mind that these parameters do not correspond to the maximum strains obtained from a conventional tensile test, due to the transverse shrinkage effects (related to the Poisson's ratio).

Azzi-Tsai-Hill 2D

This criterion uses five input parameters and outputs one indicator.

Input:

1. Axial tensile strength, $X_t > 0$
2. Axial compressive strength, $X_c > 0$
3. In-plane tensile strength, $Y_t > 0$
4. In-plane compressive strength, $Y_c > 0$
5. Transverse shear strength, $S > 0$.

The strengths have the dimension of a stress.

Output:

1. Failure indicator:

$$f_A = \sqrt{F_A(\sigma)} \text{ with } F_A(\sigma) = \frac{\sigma_{11}^2}{X^2} - \frac{|\sigma_{11}\sigma_{22}|}{X^2} + \frac{\sigma_{22}^2}{Y^2} + \frac{\sigma_{12}^2}{S^2} \tag{9-21}$$

The strengths are being taken in tension or in compression depending on the sign of their respective component $\sigma_{11} \leftrightarrow X$ & $\sigma_{22} \leftrightarrow Y$. This criterion has the same limitations as Tsai-Hill 2D, and the only difference between the two criteria is in the product σ_{11} * σ_{22}, taken here with an absolute value. This enables better failure predictions in some cases. Failure is deemed to occur when the indicator reaches or exceeds the value of 1.

Tsai-Wu 2D

This criterion uses six input parameters and outputs one indicator.

Input:

1. Axial tensile strength, $X_t > 0$
2. Axial compressive strength, $X_c > 0$
3. In-plane tensile strength, $Y_t > 0$
4. In-plane compressive strength, $Y_c > 0$
5. Transverse shear strength, $S > 0$
6. The coupled axial/in-plane strength parameter, $F > 0$. The strengths have the dimension of a stress, except the F parameter which has the dimension of the inverse of a squared stress.

Output:

1. Failure indicator:

   f_A is such that $F_A(\sigma/f) = 1$ with

$$F_A(\sigma) = \frac{\sigma_{11}^2}{X_t X_c} + \frac{\sigma_{22}^2}{Y_t Y_c} + \frac{\sigma_{12}^2}{S^2} + 2F\sigma_{11}\sigma_{22} + \left(\frac{1}{X_t} - \frac{1}{X_c}\right)\sigma_{11} + \left(\frac{1}{Y_t} - \frac{1}{Y_c}\right)\sigma_{22} \qquad (9\text{-}22)$$

The Tsai-Wu 2D criterion was designed to be more general than Tsai-Hill 2D and to be free of the main limitations of the latter model. So it is seen that tensile and compressive strengths, both axial (X_t and X_c) and perpendicular (Y_t and Y_c) all appear naturally in the criterion, and there is no need to distinguish between different cases depending on the signs of σ_{11} and σ_{22} stresses. There are also linear terms (σ_{11} and σ_{22}) that Tsai-Hill do not have. This criterion is better founded and should give better predictions than Tsai-Hill.

However, it has an important limitation. Indeed, there are several composite systems for which it is reasonable to assume that the composite does not fail in compression under service loadings. Therefore the compressive strengths X_c and Y_c are assumed to be infinite. This however renders Tsai-Wu criterion inefficient as quadratic stress terms σ_{11}^2 and σ_{22}^2 disappear. In this case, it is advised to use Tsai-Hill or Azzi-Tsai-Hill criteria instead.

Finally, as for other criteria, failure is deemed to occur according to Tsai-Wu 2D if the failure indicator reaches or exceeds the value of 1.

Tsai-Wu 2D (Strain-based)

Just like the Tsai-Hill criteria, a strain-based formulation of this criterion is available for using in Digimat-MF.

It has the following parameters:

Input:

1. Axial tensile maximum strain, $X_t > 0$
2. Axial compressive maximum strain, $X_c > 0$
3. In-plane tensile maximum strain, $Y_t > 0$
4. In-plane compressive maximum strain, $Y_c > 0$
5. Transverse shear maximum strain, $S > 0$
6. The coupled axial/in-plane maximum strain parameter, $F > 0$. All these maximum strains are dimensionless, just like strains. However, the user shall keep in mind that these parameters do not correspond to the maximum strains obtained from a conventional tensile test, due to the transverse shrinkage effects (related to the Poisson's ratio).

Output:

1. Failure indicator:

   f_A is such that $F_A(\varepsilon/f) = 1$ with

$$F_A(\varepsilon) = \frac{\varepsilon_{11}^2}{X_t X_c} + \frac{\varepsilon_{22}^2}{Y_t Y_c} + \frac{(2\varepsilon_{12})^2}{S^2} + 2F\varepsilon_{11}\varepsilon_{22} + \left(\frac{1}{X_t} - \frac{1}{X_c}\right)\varepsilon_{11} + \left(\frac{1}{Y_t} - \frac{1}{Y_c}\right)\varepsilon_{22} \qquad (9\text{-}23)$$

Tsai-Wu 3D Transversely Isotropic

This criterion assumes a material isotropy in the plane normal to axis 1 (default), 2 or 3. It has five input arguments (as it assumes identical strengths in tension and compression for shear loadings) and outputs one indicator.

Input:

1. Axial tensile strength, $X_t > 0$
2. Axial compressive strength, $X_c > 0$
3. In-plane tensile strength, $Y_t > 0$
4. In-plane compressive strength, $Y_c > 0$
5. Transverse shear strength, $S > 0$.

The strengths have the dimension of a stress.

Output:

1. Failure indicator:

   f_A is such that $F_A(\sigma/f) = 1$ with

$$F_A(\sigma) = \frac{\sigma_{11}^2}{X_t X_c} + \frac{\sigma_{22}^2 + \sigma_{33}^2}{Y_t Y_c} + \frac{\sigma_{12}^2 + \sigma_{13}^2}{S^2} + \frac{4\sigma_{23}^2}{Y_t Y_c} - \frac{\sigma_{11}\sigma_{22} + \sigma_{11}\sigma_{33}}{2X_t X_c} - \frac{2\sigma_{22}\sigma_{33}}{Y_t Y_c} + \left(\frac{1}{X_t} - \frac{1}{X_c}\right)\sigma_{11} + \left(\frac{1}{Y_t} - \frac{1}{Y_c}\right)(\sigma_{22} + \sigma_{33})$$

   when the normal to the plane of isotropy corresponds to axis 1

$$F_A(\sigma) = \frac{\sigma_{22}^2}{X_t X_c} + \frac{\sigma_{11}^2 + \sigma_{33}^2}{Y_t Y_c} + \frac{\sigma_{12}^2 + \sigma_{23}^2}{S^2} + \frac{4\sigma_{13}^2}{Y_t Y_c} - \frac{\sigma_{11}\sigma_{22} + \sigma_{22}\sigma_{33}}{2X_t X_c} - \frac{2\sigma_{11}\sigma_{33}}{Y_t Y_c} + \left(\frac{1}{X_t} - \frac{1}{X_c}\right)\sigma_{22} + \left(\frac{1}{Y_t} - \frac{1}{Y_c}\right)(\sigma_{11} + \sigma_{33})$$

   when the normal to the plane of isotropy corresponds to axis 2

$$F_A(\sigma) = \frac{\sigma_{33}^2}{X_t X_c} + \frac{\sigma_{11}^2 + \sigma_{22}^2}{Y_t Y_c} + \frac{\sigma_{13}^2 + \sigma_{23}^2}{S^2} + \frac{4\sigma_{12}^2}{Y_t Y_c} - \frac{\sigma_{22}\sigma_{33} + \sigma_{11}\sigma_{33}}{2X_t X_c} - \frac{2\sigma_{11}\sigma_{22}}{Y_t Y_c} + \left(\frac{1}{X_t} - \frac{1}{X_c}\right)\sigma_{33} + \left(\frac{1}{Y_t} - \frac{1}{Y_c}\right)(\sigma_{11} + \sigma_{22}) \quad (9\text{-}24)$$

   when the normal to the plane of isotropy corresponds to axis 3

Failure is deemed to occur when the indicator reaches or exceeds the value of 1.

Tsai-Wu 3D Transversely Isotropic (Strain based)

Just like the Tsai-Hill 3D transversely isotropic criterion, a strain-based formulation of the Tsai-Wu 3D transversely isotropic criterion is available. It assumes a material isotropy in the plane normal to axis 1 (default), 2 or 3 and has the following parameters.

Input:

1. Maximum axial tensile strain, $X_t > 0$
2. Maximum axial compressive strain, $X_c > 0$
3. Maximum in-plane tensile strain, $Y_t > 0$
4. Maximum in-plane compressive strain, $Y_c > 0$
5. Maximum transverse shear strain, $S > 0$. All these maximum strains are dimensionless, just like strains.

However, the user shall keep in mind that these parameters do not correspond to the maximum strains obtained from a conventional tensile test, due to the transverse shrinkage effects (related to the Poisson's ratio).

Output:

1. Failure indicator:

   f_A is such that $F_A(\varepsilon/f) = 1$ with

$$F_A(\varepsilon) = \frac{\varepsilon_{11}^2}{X_t X_c} + \frac{\varepsilon_{22}^2 + \varepsilon_{33}^2}{Y_t Y_c} + \frac{(2\varepsilon_{12})^2 + (2\varepsilon_{13})^2}{S^2} + \frac{(2\varepsilon_{23})^2}{Y_t Y_c} - \frac{\varepsilon_{11}\varepsilon_{22} + \varepsilon_{11}\varepsilon_{33}}{2X_t X_c} - \frac{2\varepsilon_{22}\varepsilon_{33}}{Y_t Y_c} + \left(\frac{1}{X_t} - \frac{1}{X_c}\right)\varepsilon_{11} + \left(\frac{1}{Y_t} - \frac{1}{Y_c}\right)(\varepsilon_{22} + \varepsilon_{33})$$

   when the normal to the plane of isotropy corresponds to axis 1

$$F_A(\varepsilon) = \frac{\varepsilon_{22}^2}{X_t X_c} + \frac{\varepsilon_{11}^2 + \varepsilon_{33}^2}{Y_t Y_c} + \frac{\varepsilon_{12}{}^2 + \varepsilon_{23}{}^2}{S^2} + \frac{4\varepsilon_{13}^2}{Y_t Y_c} - \frac{\varepsilon_{11}\varepsilon_{22} + \varepsilon_{22}\varepsilon_{33}}{2X_t X_c} - \frac{2\varepsilon_{11}\varepsilon_{33}}{Y_t Y_c} + \left(\frac{1}{X_t} - \frac{1}{X_c}\right)\varepsilon_{22} + \left(\frac{1}{Y_t} - \frac{1}{Y_c}\right)(\varepsilon_{11} + \varepsilon_{33})$$

   when the normal to the plane of isotropy corresponds to axis 2

$$F_A(\varepsilon) = \frac{\varepsilon_{33}^2}{X_t X_c} + \frac{\varepsilon_{11}^2 + \varepsilon_{22}^2}{Y_t Y_c} + \frac{\varepsilon_{13}^2 + \varepsilon_{23}^2}{S^2} + \frac{4\varepsilon_{12}^2}{Y_t Y_c} - \frac{\varepsilon_{22}\varepsilon_{33} + \varepsilon_{11}\varepsilon_{33}}{2X_t X_c} - \frac{2\varepsilon_{11}\varepsilon_{22}}{Y_t Y_c} + \left(\frac{1}{X_t} - \frac{1}{X_c}\right)\varepsilon_{33} + \left(\frac{1}{Y_t} - \frac{1}{Y_c}\right)(\varepsilon_{11} + \varepsilon_{22}) \tag{9-25}$$

   when the normal to the plane of isotropy corresponds to axis 3

Failure is deemed to occur when the indicator reaches or exceeds the value of 1.

Tsai-Wu 3D Orthotropic

This criterion has twelve input arguments and outputs one indicator.

Input:

1. Tensile strength 1, $X_t > 0$
2. Tensile strength 2, $Y_t > 0$
3. Tensile strength 3, $Z_t > 0$
4. Compressive strength 1, $X_c > 0$
5. Compressive strength 2, $Y_c > 0$
6. Compressive strength 3, $Z_c > 0$

7. Shear strength 12, $S_{xy} > 0$
8. Shear strength 23, $S_{yz} > 0$
9. Shear strength 31, $S_{zx} > 0$
10. Reduced integration term 12, $F^*_{xy} \in [-1, 1]$
11. Reduced integration term 23, $F^*_{yz} \in [-1, 1]$
12. Reduced integration term 31, $F^*_{zx} \in [-1, 1]$

The strengths have the dimension of a stress.

Output:

1. Failure indicator:

$$F_A(\sigma) = \frac{\sigma_{11}^2}{X_t X_c} + \frac{\sigma_{22}^2}{Y_t Y_c} + \frac{\sigma_{33}^2}{Z_t Z_c} + \frac{\sigma_{12}^2}{S_{xy}^2} + \frac{\sigma_{23}^2}{S_{yz}^2} + \frac{\sigma_{31}^2}{S_{zx}^2} + 2F_{xy}\sigma_{11}\sigma_{22} + 2F_{yz}\sigma_{22}\sigma_{33} + 2F_{zx}\sigma_{11}\sigma_{33} + \left(\frac{1}{X_t} - \frac{1}{X_c}\right)\sigma_{11} + \left(\frac{1}{Y_t} - \frac{1}{Y_c}\right)\sigma_{22} + \left(\frac{1}{Z_t} - \frac{1}{Z_c}\right)\sigma_{33} \quad (9\text{-}26)$$

where the interaction terms F_{AB} are defined from the reduced one:

$$F_{AB} = \frac{F^*_{AB}}{\sqrt{A_t A_c B_t B_c}} \quad (9\text{-}27)$$

From failure mode considerations, a negative value should usually be chosen for the reduced interaction terms. Failure is deemed to occur when the indicator reaches or exceeds the value of 1.

Tsai-Wu 3D Orthotropic (Strain based)

This criterion has twelve input arguments and outputs one indicator.

Input:

1. Maximum tensile strain 1, $X_t > 0$
2. Maximum tensile strain 2, $Y_t > 0$
3. Maximum tensile strain 3, $Z_t > 0$
4. Maximum compressive strain 1, $X_c > 0$
5. Maximum compressive strain 2, $Y_c > 0$
6. Maximum compressive strain 3, $Z_c > 0$

7. Maximum shear strain 12, $S_{xy} > 0$
8. Maximum shear strain 23, $S_{yz} > 0$
9. Maximum shear strain 31, $S_{zx} > 0$
10. Reduced integration term 12, $F^*_{xy} \in [-1, 1]$
11. Reduced integration term 23, $F^*_{yz} \in [-1, 1]$
12. Reduced integration term 31, $F^*_{zx} \in [-1, 1]$

Output:

1. Failure indicator:

$$F_A(\varepsilon) = \frac{\varepsilon_{11}^2}{X_t X_c} + \frac{\varepsilon_{22}^2}{Y_t Y_c} + \frac{\varepsilon_{33}^2}{Z_t Z_c} + \frac{(2\varepsilon_{12})^2}{S_{xy}^2} + \frac{(2\varepsilon_{23})^2}{S_{yz}^2} + \frac{(2\varepsilon_{31})^2}{S_{zx}^2} + 2F_{xy}\varepsilon_{11}\varepsilon_{22} + 2F_{yz}\varepsilon_{22}\varepsilon_{33} + 2F_{zx}\varepsilon_{11}\varepsilon_{33} + \left(\frac{1}{X_t} - \frac{1}{X_c}\right)\varepsilon_{11} + \left(\frac{1}{Y_t} - \frac{1}{Y_c}\right)\varepsilon_{22} + \left(\frac{1}{Z_t} - \frac{1}{Z_c}\right)\varepsilon_{33} \quad (9\text{-}28)$$

   where the interaction terms F_{AB} are defined from the reduced one:

$$F_{AB} = \frac{F^*_{AB}}{\sqrt{A_t A_c B_t B_c}} \quad (9\text{-}29)$$

From failure mode considerations, a negative value should usually be chosen for the reduced interaction terms. Failure is deemed to occur when the indicator reaches or exceeds the value of 1.

Tsai-Wu 3D

This indicator uses twenty-seven input parameters and outputs one indicator.

Input:

1. The 6 components of a second order symmetric tensor, H_{ij}.
2. The 21 components of a fourth order symmetric tensor, G_{ijkl}. Units are the inverse of a stress for the H tensor terms and the inverse of a squared stress for the G tensor terms.

Output:

1. Failure indicator, with summation over the indices

   f_A is such that $F_A(\sigma/f) = 1$ with $F_A(\sigma) = H_{ij}\sigma_{ij} + G_{ijkl}\sigma_{ij}\sigma_{kl}$ (9-30)

Failure is deemed to occur if the failure indicator reaches or exceeds the value of 1.

Multi-component 2D

This indicator uses five input parameters and outputs five indicators.

Input:

1. Axial tensile strength, $X_t > 0$
2. Axial compressive strength, $X_c > 0$
3. In-plane tensile strength, $Y_t > 0$
4. In-plane compressive strength, $Y_c > 0$
5. Transverse shear strength, $S > 0$.

The strengths have the dimension of a stress.

Output:

1. Tensile failure indicator in 1-direction

$$f_A = F_A(\sigma) \text{ with } F_A(\sigma) = \frac{\sigma_{11}}{X_t} \text{ if } \sigma_{11} \geq 0, \text{ 0 otherwise} \tag{9-31}$$

2. Compressive failure indicator in 1-direction

$$f_B = F_B(\sigma) \text{ with } F_B(\sigma) = -\frac{\sigma_{11}}{X_c} \text{ if } \sigma_{11} < 0, \text{ 0 otherwise} \tag{9-32}$$

3. Tensile failure indicator in 2-direction

$$f_C = F_C(\sigma) \text{ with } F_C(\sigma) = \frac{\sigma_{22}}{Y_t} \text{ if } \sigma_{22} \geq 0, \text{ 0 otherwise} \tag{9-33}$$

4. Compressive failure indicator in 2-direction

$$f_D = F_D(\sigma) \text{ with } F_D(\sigma) = -\frac{\sigma_{22}}{Y_c} \text{ if } \sigma_{22} < 0, \text{ 0 otherwise} \tag{9-34}$$

5. Transverse shear failure indicator (in 1-2 plane)

$$f_E = F_E(\sigma) \text{ with } F_E(\sigma) = \frac{|\sigma_{12}|}{S} \tag{9-35}$$

Failure happens if one of the failure indicators reaches or exceeds the critical value of 1.

This failure indicator can be used with a multi-components 2D progressive failure model (see subsection Multi-components 2D Damage).

Multi-component 3D

This indicator uses nine input parameters and outputs nine indicators.

Input:

1. Axial tensile strength, $X_t > 0$
2. Axial compressive strength, $X_c > 0$
3. In-plane tensile strength, $Y_t > 0$
4. In-plane compressive strength, $Y_c > 0$
5. Out-of-plane tensile strength, $Z_t > 0$
6. In-Out compressive strength, $Z_c > 0$
7. Shear 12 strength, $S_{xy} > 0$
8. Shear 13 strength, $S_{xz} > 0$
9. Shear 23 strength, $S_{yz} > 0$, $S > 0$

The strengths have the dimension of a stress.

Output:

1. Tensile failure indicator in 1-direction

$$f_A = F_A(\sigma) \text{ with } F_A(\sigma) = \frac{\sigma_{11}}{X_t} \text{ if } \sigma_{11} \geq 0, \text{ 0 otherwise} \tag{9-36}$$

2. Compressive failure indicator in 1-direction

$$f_B = F_B(\sigma) \text{ with } F_B(\sigma) = -\frac{\sigma_{11}}{X_c} \text{ if } \sigma_{11} < 0, \text{ 0 otherwise} \tag{9-37}$$

3. Tensile failure indicator in 2-direction

$$f_C = F_C(\sigma) \text{ with } F_C(\sigma) = \frac{\sigma_{22}}{Y_t} \text{ if } \sigma_{22} \geq 0, \text{ 0 otherwise} \tag{9-38}$$

4. Compressive failure indicator in 2-direction

$$f_D = F_D(\sigma) \text{ with } F_D(\sigma) = -\frac{\sigma_{22}}{Y_c} \text{ if } \sigma_{22} < 0, \text{ 0 otherwise} \tag{9-39}$$

5. Tensile failure indicator in 3-direction

$$f_C = F_C(\sigma) \text{ with } F_C(\sigma) = \frac{\sigma_{33}}{Z_t} \text{ if } \sigma_{33} \geq 0,\ 0 \text{ otherwise} \tag{9-40}$$

6. Compressive failure indicator in 3-direction

$$f_D = F_D(\sigma) \text{ with } F_D(\sigma) = -\frac{\sigma_{33}}{Z_c} \text{ if } \sigma_{33} < 0,\ 0 \text{ otherwise} \tag{9-41}$$

7. Shear 12 failure indicator

$$f_E = F_E(\sigma) \text{ with } F_E(\sigma) = \frac{|\sigma_{12}|}{S_{xy}} \tag{9-42}$$

8. Shear 13 failure indicator

$$f_E = F_E(\sigma) \text{ with } F_E(\sigma) = \frac{|\sigma_{13}|}{S_{xz}} \tag{9-43}$$

9. Shear 23 failure indicator

$$f_E = F_E(\sigma) \text{ with } F_E(\sigma) = \frac{|\sigma_{23}|}{S_{yz}} \tag{9-44}$$

Failure happens if one of the failure indicators reaches or exceeds the critical value of 1.

Hashin-Rotem 2D

This indicator uses five input parameters and outputs four indicators.

Input:

1. Axial tensile strength, $X_t > 0$
2. Axial compressive strength, $X_c > 0$
3. In-plane tensile strength, $Y_t > 0$
4. In-plane compressive strength, $Y_c > 0$
5. Transverse shear strength, $S > 0$

The strengths have the dimension of a stress.

Output:

1. Tensile failure indicator in 1-direction

$$f_A = F_A(\sigma) \text{ with } F_A(\sigma) = \frac{\sigma_{11}}{X_t} \text{ if } \sigma_{11} \geq 0,\ 0 \text{ otherwise} \tag{9-45}$$

2. Compressive failure indicator in 1-direction

$$f_B = F_B(\sigma) \text{ with } F_B(\sigma) = -\frac{\sigma_{11}}{X_c} \text{ if } \sigma_{11} < 0,\ 0 \text{ otherwise} \tag{9-46}$$

3. Coupled tensile/shear failure indicator

$$f_C = \sqrt{F_C(\sigma)} \text{ with } F_C(\sigma) = \frac{\sigma_{22}^2}{Y_t^2} + \frac{\sigma_{12}^2}{S^2} \text{ if } \sigma_{22} \geq 0,\ 0 \text{ otherwise} \tag{9-47}$$

4. Coupled compressive/shear failure indicator

$$f_D = \sqrt{F_D(\sigma)} \text{ with } F_D(\sigma) = \frac{\sigma_{22}^2}{Y_c^2} + \frac{\sigma_{12}^2}{S^2} \text{ if } \sigma_{22} < 0,\ 0 \text{ otherwise} \tag{9-48}$$

Failure happens if one of the failure indicators reaches or exceeds the critical value of 1.

This failure indicator can be used with a Matzenmiller 2D progressive failure model (see subsection Matzenmiller 2D Damage).

Hashin 2D

This indicator comprises six parameters and outputs four variables.

Input:

1. Axial tensile strength, $X_t > 0$
2. Axial compressive strength, $X_c > 0$
3. In-plane tensile strength, $Y_t > 0$
4. In-plane compressive strength, $Y_c > 0$
5. Transverse shear strength, in the plane (1,2), $S > 0$
6. The in-plane shear strength, in the plane (2,3), $SI > 0$.

The strengths have the dimension of a stress.

Output

1. Coupled tensile/shear failure indicator in 1-direction

$$f_A = F_A(\sigma) \text{ with } F_A(\sigma) = \frac{\sigma_{11}^2}{X_t^2} + \frac{\sigma_{12}^2}{S^2} \text{ if } \sigma_{11} \geq 0,\ 0 \text{ otherwise} \tag{9-49}$$

2. Compressive failure indicator

$$f_B = F_B(\sigma) \text{ with } F_B(\sigma) = -\frac{\sigma_{11}}{X_c} \text{ if } \sigma_{11} < 0, \text{ 0 otherwise} \tag{9-50}$$

3. Coupled tensile/shear failure indicator in 2-direction

$$f_C = \sqrt{F_C(\sigma)} \text{ with } F_C(\sigma) = \frac{\sigma_{22}^2}{Y_t^2} + \frac{\sigma_{12}^2}{S^2} \text{ if } \sigma_{22} \geq 0, \text{ 0 otherwise} \tag{9-51}$$

4. Coupled compressive/shear/in-plane shear failure indicator in 2-direction

   f_D is such that $F_D(\sigma/f) = 1$ with

$$F_D(\sigma) = \frac{\sigma_{22}^2}{4S_I^2} + \frac{\sigma_{12}^2}{S^2} + \left[\left(\frac{Y_c}{2S_I}\right)^2 - 1\right]\frac{\sigma_{22}^2}{Y_c} \text{ if } \sigma_{22} < 0, \text{ 0 otherwise} \tag{9-52}$$

Failure happens if one of the failure indicators reaches the critical value of 1.

This failure indicator can be used with a Matzenmiller 2D progressive failure model (see subsection Matzenmiller 2D Damage).

Hashin 3D

This indicator comprises six parameters and outputs four variables.

Input:

1. Axial tensile strength, $X_t > 0$
2. Axial compressive strength, $X_c > 0$
3. In-plane tensile strength, $Y_t > 0$
4. In-plane compressive strength, $Y_c > 0$
5. Transverse shear strength, in the plane (1,2), $S > 0$
6. The in-plane shear strength, in the plane (2,3), $SI > 0$.

The strengths have the dimension of a stress.

Output:

1. Coupled tensile/shear failure indicator in 1-direction

$$f_A = \sqrt{F_A(\sigma)} \text{ with } F_A(\sigma) = \frac{\sigma_{11}^2}{X_t^2} + \frac{\sigma_{12}^2 + \sigma_{13}^2}{S^2} \text{ if } \sigma_{11} \geq 0, \text{ 0 otherwise} \tag{9-53}$$

2. Compressive failure indicator

$$f_B = F_B(\sigma) \text{ with } F_B(\sigma) = -\frac{\sigma_{11}}{X_c} \text{ if } \sigma_{11} < 0,\ 0 \text{ otherwise} \quad (9\text{-}54)$$

3. Coupled tensile/shear failure indicator in 2-direction

$$f_C = \sqrt{F_C(\sigma)} \text{ with } F_C(\sigma) = \frac{(\sigma_{22}+\sigma_{33})^2}{Y_t^2} + \frac{\sigma_{12}^2+\sigma_{13}^2}{S^2} + \frac{\sigma_{23}^2-\sigma_{22}\sigma_{33}}{S_I^2} \quad (9\text{-}55)$$

   if $(\sigma_{22}+\sigma_{33}) \geq 0,\ 0$ otherwise

4. Coupled compressive/shear/in-plane shear failure indicator in 2-direction

   f_D is such that $F_D(\sigma/f) = 1$ with

$$F_D(\sigma) = \frac{(\sigma_{22}+\sigma_{33})^2}{4S_I^2} + \frac{\sigma_{12}^2+\sigma_{13}^2}{S^2} + \frac{\sigma_{23}^2-\sigma_{22}\sigma_{33}}{S_I^2} + \left[\left(\frac{Y_c}{2S_I}\right)^2 - 1\right]\frac{\sigma_{22}+\sigma_{33}}{Y_c} \quad (9\text{-}56)$$

   if $(\sigma_{22}+\sigma_{33}) < 0,\ 0$ otherwise

Failure happens if one of the failure indicators reaches or exceeds the critical value of 1.

This failure indicator can be used with a Matzenmiller 3D progressive failure model (see subsection Matzenmiller 3D Damage).

SIFT

This indicator comprises five parameters and outputs five variables.

Input:

1. Critical J_1 tensile strain, $J_{1t}^{crit} > 0$
2. Critical J_1 compressive strain, $\left|J_{1c}^{crit}\right| > 0$
3. Critical J_2 tensile strain, $J_{2t}^{crit} > 0$
4. Critical J_2 compressive strain, $\left|J_{2c}^{crit}\right| > 0$
5. Critical I_2 strain, $I_2^{crit} > 0$

Output:

The SIFT failure criterion is based on the three following strain invariants:

$$\begin{aligned} J_1 &= \varepsilon_{11} + \varepsilon_{22} + \varepsilon_{33} = \varepsilon_{vol} \\ J_2 &= \varepsilon_{11}\varepsilon_{22} + \varepsilon_{11}\varepsilon_{33} + \varepsilon_{22}\varepsilon_{33} - \varepsilon_{12}^2 - \varepsilon_{13}^2 - \varepsilon_{23}^2 \text{ and} \\ I_2 &= J_1^2 - 3J_2 = \varepsilon_{eq}^2 \end{aligned} \tag{9-57}$$

The corresponding failure indicators are:

1. J_1 (volumic strain) tensile failure indicator

$$f_A = F_A(\sigma) \text{ with } F_A(\sigma) = \frac{J_1}{J_{1t}^{crit}} \text{ if } J_1 > 0, \text{ 0 otherwise} \tag{9-58}$$

2. J_1 (volumic strain) compressive failure indicator

$$f_B = F_B(\sigma) \text{ with } F_B(\sigma) = -\frac{J_1}{J_{1c}^{crit}} \text{ if } J_1 < 0, \text{ 0 otherwise} \tag{9-59}$$

3. J_2 tensile failure indicator

$$f_C = \sqrt{F_C(\sigma)} \text{ with } F_C(\sigma) = \frac{J_2}{J_{2t}^{crit}} \text{ if } J_2 > 0, \text{ 0 otherwise} \tag{9-60}$$

4. J_2 compressive failure indicator

$$f_D = \sqrt{F_D(\sigma)} \text{ with } F_D(\sigma) = -\frac{J_2}{\left|J_{2c}^{crit}\right|} \text{ if } J_2 < 0, \text{ 0 otherwise} \tag{9-61}$$

5. I_2 (equivalent strain) failure indicator

$$f_E = \sqrt{F_E(\sigma)} \text{ with } F_E(\sigma) = \frac{I_2}{I_2^{crit}} \tag{9-62}$$

Failure happens if one of the failure indicators reaches or exceeds the critical value of 1.

Remark: As this criterion is based on strain invariants, the output values will be independent on the axis system chosen by the user.

Christensen

This criterion has two input arguments and outputs two indicators.

Input:

1. Axial tensile strength, $X_t > 0$

2. Axial compressive strength, $X_c > 0$.

The strengths have the dimension of a stress.

output:

1. Main failure indicator:

   f_A is such that $F_A(\sigma/f) = 1$ with

$$F_A(\sigma) = \left(\frac{1}{X_t} - \frac{1}{X_c}\right)(\sigma_{11} + \sigma_{22} + \sigma_{33}) + \frac{1}{X_t X_c}\left\{\frac{1}{2}[(\sigma_{11} - \sigma_{22})^2 + (\sigma_{22} - \sigma_{33})^2 + (\sigma_{33} - \sigma_{11})^2] + 3(\sigma_{12}^2 + \sigma_{23}^2 + \sigma_{31}^2)\right\} \quad (9\text{-}63)$$

2. Brittle failure indicator:

   $f_B = \sqrt{F_B(\sigma)}$ with $F_B(\sigma) = \sigma_1 - X_t$

where σ_1 is the largest principal stress. This second indicator is only valid in the brittle domain where $2X_t \leq X_c$.

The Christensen criterion is used for the failure of isotropic materials. It differentiates between ductile yielding (1st indicator) and brittle failure (2nd indicator), inducing a pressure dependency by the explicit presence of the hydrostatic stress in the first indicator. It reduces to the von Mises criterion when $X_t = X_c$. However, no attempt is made to distinguish between ductile and brittle responses. In this situation, failure is considered in an inclusive sense encompassing both yielding and complete rupture.

Camanho

This criterion has eight input arguments and outputs four indicators. This failure indicator is designed for composites reinforced with unidirectional continuous fibers. The outputs are related to four different failure mechanisms: longitudinal failure in tension, longitudinal failure in compression, transverse failure at 90 degrees and transverse failure at an angle smaller than 90 degrees.

Inputs:

1. Longitudinal tension strength, $X_T > 0$
2. Longitudinal compression strength, $X_C > 0$
3. Transverse tension strength, $Y_T > 0$
4. Fracture toughness in transverse tension, $G_{YT} > 0$

5. Transverse compression strength, $Y_C > 0$
6. In-plane shear strength, $S_L > 0$
7. Fracture toughness in in-plane shear, $G_{SL} > 0$
8. Transverse compression fracture angle, α_0

The strengths have the dimension of a stress, the fracture toughnesses have the dimension of an energy per unit area and the angle needs to be entered in radians.

Outputs:

1. Longitudinal failure in tension (due to fiber failure)

$$f_A = F_A(\varepsilon, \sigma) \text{ with } F_A(\varepsilon, \sigma) = \begin{cases} \frac{E_1}{X_T}\varepsilon_{11} & \text{if } \sigma_{11} \geq 0 \\ 0 & \text{otherwise} \end{cases} \tag{9-64}$$

2. Longitudinal failure in compression (due to fiber kinking)

$$f_B = F_B(\varepsilon, \sigma) \text{ with } F_B(\varepsilon, \sigma) = \begin{cases} \frac{\langle \left|\sigma_{12}^m\right| + \eta^L \sigma_{22}^m \rangle}{S_L} & \text{if } \sigma_{11} < 0 \\ 0 & \text{otherwise} \end{cases} \tag{9-65}$$

3. Transverse failure at 90 degrees (due to matrix failure under transverse tensile or shear dominated loads)

$f_C = F_C(\varepsilon, \sigma)$ with

$$F_C(\varepsilon, \sigma) = \begin{cases} \sqrt{\left(1 - \frac{G_{YT}}{G_{SL}}\right)\frac{\sigma_{22}}{Y_T} + \frac{G_{YT}}{G_{SL}}\left(\frac{\sigma_{22}}{Y_T}\right)^2 + \left(\frac{\sigma_{12}}{S_L}\right)^2} & \text{if } \sigma_{22} \geq 0 \\ \frac{1}{S_L}\langle \left|\sigma_{12}\right| + \eta^L \sigma_{22} \rangle & \text{otherwise} \end{cases} \tag{9-66}$$

4. Transverse failure at an angle 0 (due to matrix failure under transverse compression dominated loads)

$$f_D = F_D(\varepsilon, \sigma) \text{ with } F_D(\varepsilon, \sigma) = \begin{cases} \sqrt{\left(\frac{T_{eff}^T}{S_T}\right)^2 + \left(\frac{T_{eff}^L}{S_L}\right)^2} & \text{if } \sigma_{22} < 0 \\ 0 & \text{otherwise} \end{cases} \tag{9-67}$$

where:

$$\eta^{L} = -\frac{[S_L \cos(2\alpha_0)]}{[Y_C \cos^2\alpha_0]}$$

$$\eta^{T} = -\frac{1}{\tan(2\alpha_0)}$$

$$\theta = \arctan\left[\frac{-|\sigma_{12}|}{(\sigma_{22}\sin(\alpha_0))}\right]$$

$$S_T = Y_C \cos\alpha_0 [\sin\alpha_0 + \cos\alpha_0 / \tan(2\alpha_0)]$$

$$\tan\varphi^{C} = \frac{\left[1 - \sqrt{1 - 4\left(\frac{S_L}{X_C} + \eta^{L}\right)\frac{S_L}{X_C}}\right]}{\left[2\left(\frac{S_L}{X_C} + \eta^{L}\right)\right]}$$

$$\sigma_{12}^{m} = (\sigma_{22} - \sigma_{11})\sin\varphi^{C}\cos\varphi^{C} + |\sigma_{12}|\left(\cos^2\varphi^{c} - \sin^2\varphi^{c}\right)$$

$$\sigma_{22}^{m} = \sigma_{11}\sin^2\varphi^{c} + \sigma_{22}\cos^2\varphi^{c} + 2|\sigma_{12}|\sin\varphi^{c}\cos\varphi^{c}$$

$$\tau_{eff}^{T} = \langle -\sigma_{22}\cos\alpha_0 [\sin\alpha_0 - \eta^{T}\cos\alpha_0 \cdot \cos\theta] \rangle$$

$$\tau_{eff}^{L} = \langle \cos\alpha_0 (|\sigma_{12}| + \eta^{L}\sigma_{22}\cos\alpha_0 \cdot \sin\theta) \rangle$$

Failure happens if one of the failure indicators reaches or exceeds the critical value of 1.

Accumulated Plastic Strain

This criterion has a function as input argument and outputs one indicator.

The input is a series of value pairs, describing the evolution of the critical accumulated plastic strain versus the stress triaxiality ratio. The values of the triaxiality ratio must be sorted in strictly increasing order. The critical accumulated plastic strain is evaluated as a function of the triaxiality ratio with a piecewise linear interpolation within the range values of the triaxiality ratio.

Two different extrapolations are used at the upper and lower bound of triaxiality. The critical accumulated plastic strain is constantly extrapolated for triaxiality which are bigger than the upper bound. The critical accumulated plastic strain is linearly extrapolated for triaxiality which are lower than the lower bound.

All the input quantities are dimensionless.

Output:

Main failure indicator:

$$F_A(\eta, p) = \frac{p}{p_{critical}(\eta)} \text{ with } \begin{cases} \eta = \dfrac{-\sigma_H}{\sigma_{eq}} = \dfrac{tr(\sigma)}{3\sqrt{3J_2(\sigma)}} & \text{the triaxiality ratio} \\ p & \text{the accumulated plastic strain} \end{cases} \quad (9\text{-}68)$$

The accumulated plastic strain criterion is meant to be used with (visco-)elasto-(visco-)plastic materials; typically, on the matrix phase of the composite. This failure indicator cannot be used at the macroscopic (Composite) level, except when homogenization is switched off, since the accumulated plastic strain is not defined at this level.

In some conditions, this failure indicator can also be expressed as an incremental formulation:

$$\Delta F_A(eta, p) = \frac{\Delta p}{p_{critical}} \quad (9\text{-}69)$$

which is accumulated over time increments. This formulation yields a significantly better descriptions (w.r.t. the total formulation) in the case of non-monotonous, non-proportional, or non-isothermal loadings. In Digimat-CAE and Digimat-RP, it is available with the Micro solution procedure only.

User-defined

Digimat enables the user to implement its own failure indicators, in the form of an external library (a DLL file on Windows or a shared object on Linux), and to use it like any other Digimat failure indicator.

Warning: This capability is dedicated to failure experts having basic knowledge of development tools, methods and languages. Users are cautioned that the implementation of realistic user-defined failure indicators requires extensive development and testing, as well as significant expertize on failure topics. Hexagon shall not be liable for the consequential damages resulting from an incorrect implementation, testing or application of a customer user-defined failure indicator. However, we provide support for this capability (digimat.support@hexagon.com), as well as useful examples and guidelines in the Frequently Asked Questions below.

Usage

The usage of user-defined failure indicator in Digimat requires the definition of the following settings in the Digimat Graphical User Interface (see Figure 9-2):

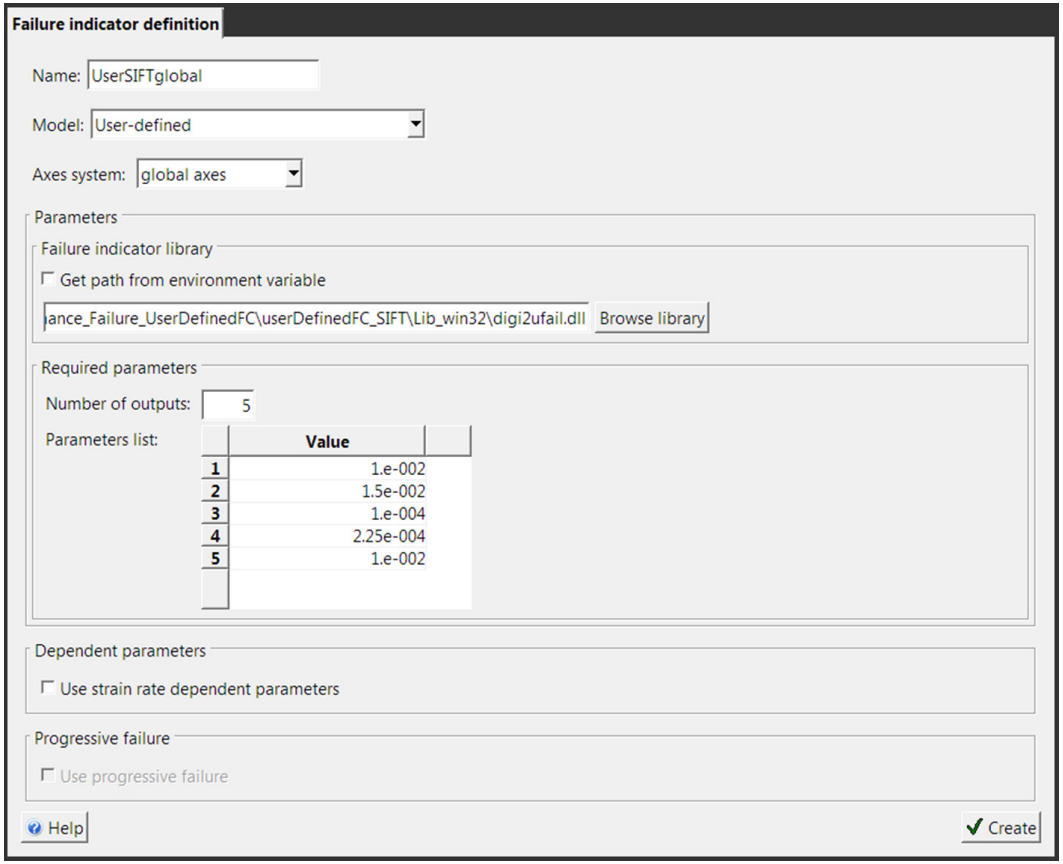


Figure 9-2 Definition of a user-defined failure indicator in Digimat-MF GUI.

- The **path to the Dynamic Library** packaging the user subroutine (a DLL file on Windows or a shared object on Linux). This path can be defined either:
  - By explicitly defining this path through the **Browse library** button in the GUI;
  - Using an environment variable named `DIGIMAT2USUB_SHARED_LIBS`, which should give the path to a folder containing the DLL `digi2ufail.dll` on Windows or the shared object `digi2ufail.so` on Linux. This enables the user to define a configuration-independent analysis file, which is especially useful for preparing a Digimat analysis file to be run on another computer.
- The **number of failure indicators** that will be returned by the user subroutine. The minimal value for this setting is one. If several outputs are provided, Digimat will use the same conventions as for classical failure criteria (see above).

- The **number of (internal) history variables** that are required the user subroutine. The minimal value for this setting is zero, and shall remain zero in most implementation. Note that history variables are only available in some configurations (see subsection User-defined Failure Indicator).
- **A list of real-valued parameters** that will be passed to the user subroutine (typically, material strengths). At least one parameter must be defined (use a dummy value if necessary).
- **The number of failure indicators** that will be returned by the user subroutine. The minimal value for this setting is one. If several outputs are provided, Digimat will use the same conventions as for classical failure criteria (see above).

Strain-rate, temperature and user-variable dependencies can be used with user-defined failure indicators.

This failure indicator can be used with a General Anisotropic progressive failure model (see subsection General Anisotropic Damage).

Implementation

The user implementation is fully customizable, but must implement the following function:

```
computeUserFailureCriterion (
      int* errorFlag , double * criterionOutputs ,
      const int* numberOfCriterionOutputs ,
      const double * inputValues , const int* numberOfInputs ,
      const double * parameters , const int* numberOfParameters ,
      const char * criterionName ,
      const double * options , const int* numberOfOptions );
      double * stateVariables , const int* nStateVariables );
```

Each time it is necessary, the Digimat-MF kernel will call the computeUserFailureCriterion function with preinitialized arguments (the user does not need to manage variables allocations). This Digimat capability is not linked to any particular programming language: it can be implemented in C, C++, Fortran... This is the reason why the parameters are passed by pointers.

The two first parameters are output parameters and must be updated by the subroutine:

- errorFlag is interpreted as an error code by the Digimat-MF kernel:
  - a null value (0) means success
  - a negative value (<0) means success with warning
  - a positive value (>0) means error.

  Warning and error codes will generate a message in the Digimat log file, but will not stop the analysis.
- `criterionOutputs` is an array containing all the numerical outputs of the failure indicator; it must be updated by the user subroutine

The two last parameters are input/output parameters:

- `stateVariables` is an array of double-valued history variables. Its values are initialized at the beginning of the timestep, and shall be updated by the user subroutine.
- `nStateVariables` is the size of the stateVariables array (input only).

Remark: Failure history variables are only available for a few configurations (see subsection Failure). For all other configurations, stateVariables is a null pointer, and the value of nStateVariables is equal to zero. If the user implementation requires history variables, it is recommended to datacheck those parameters.

All the others are informative values defined by Digimat from the analysis settings, and cannot be modified.

- `numberOfCriterionOutputs` is the size of the criterionOutputs array, i.e., the number of failure indicators computed by the user subroutine.
- `inputValues` is an array containing all the material state inputs necessary for the computation of the failure criterion.The number of inputs depend on the context. In Digimat-MF, it contains the values of the stress and strain tensor components, and the accumulated plastic strain:

  $\{\sigma_{11}, \sigma_{22}, \sigma_{33}, \sigma_{12}, \sigma_{23}, \sigma_{13}, \varepsilon_{11}, \varepsilon_{22}, \varepsilon_{33}, \varepsilon_{12}, \varepsilon_{23}, \varepsilon_{13}, p, T, \Delta t\}$

  In the case of standard failure criteria without progressive failure, it also contains the stress and strain tensor components, the accumulated plastic strain, and the temperature at the beginning of the timestep more information may be added in future versions of Digimat.

  Remark: The stress and strain tensors are already thrown into the axis system (global, local or principal) specified in Digimat.
- `nInputs` is the size of the `inputValues` array.
- `parameters` is an array containing all the parameters necessary for the computation of the failure criterion; these parameters can have a physical meaning (typically, material strengths), or can be used as flags (to choose between several models, for example). If dependencies are defined in Digimat for these parameters, the values passed to the user subroutine will account for them; consequently, these parameters may vary along the analysis.
- `numberOfParameters` is the size of the parameters array.
- `criterionName` is a null-terminated chain of characters corresponding to the name of the criterion defined in Digimat. This argument is only informative, and can be used as a flag (to choose between several models, for example).
- `options` is an array of additional options. This argument can be ignored for now: it is a purely prospective argument, only meant to let more information be passed in future versions of Digimat, while preserving compatibility. It the current Digimat version, a null pointer is passed.
- `numberOfOptions` is the size of the options array; it is null in the current Digimat version.

Frequently Asked Questions

A list of the most Frequently Asked Questions for this capability is provided in the respective Guidelines section (see subsection User-defined Failure Indicator).

Failure Criterion Definition & Assignment

This page introduces the procedure to assign a failure criterion in a Digimat analysis. This procedure consists of two main steps:

1. Definition of a failure indicator.
2. Assignment of this failure indicator to a given material level.

The page continues with notes on the Kelly-Tyson tensile strength estimator for composite materials and some information about element deletion and the stopping point in coupled analyses.

Definition of a Failure Indicator

The failure indicator definition window can be accessed by a left click on a failure indicator item in the Digimat tree (only if failure indicators already exist in the current analysis), or by right clicking on the failure item in the Digimat tree and selecting "Add failure indicator" in the context menu that pops up. It is also possible through this context menu to load a failure indicator that was previously saved to a Digimat Failure Indicator (`*.dfi`) file.

The definition of a failure indicator in Digimat requires the selection of a given failure indicator model and the specification of the axis system in which the stress or strain tensor components are expressed and of the failure indicators parameters. A name should also be given to the defined failure indicator.

For more information on the available failure indicators in Digimat, see section Failure Indicators.

Figure 9-3 illustrates the failure indicator definition window. Setting dependencies on the failure indicator parameters can be done via the second tab in this window. See section Setting Dependencies for more information.

The stress and strain tensor components can be expressed in three different axis systems.

- The global axes:
  - in Digimat-MF, this corresponds to the axis system with respect to which the loadings and the inclusion orientation are defined
  - in a Digimat-CAE analysis, this corresponds to the element axes, i.e., the model axes for solid elements, the local shell axes for shell elements, or user-defined axes if any are defined in the FE model.

Failure indicators defined in the global axis system should only be assigned at the composite level.

- The local axes:
  - fixed orientation: these correspond to the axis of revolution of the inclusions completed by 2 perpendicular axes
  - distributed orientation: they correspond to the eigenvectors of the orientation tensor defining the inclusion orientation.

These automatically determined axes can be overridden by user-defined ones defined via the axes manager.

For a distributed orientation, failure indicators defined in the local axis system should only be assigned at the inclusion phase level, unless the FPGF method is used.

- The tensor's principal axes – these correspond to the eigenvectors of the stress or strain tensor considered in the failure indicator.

Note that the Kelly-Tyson estimator can be used to estimate the tensile strength parameter required by some of the available failure indicators, e.g., **Stress component**.

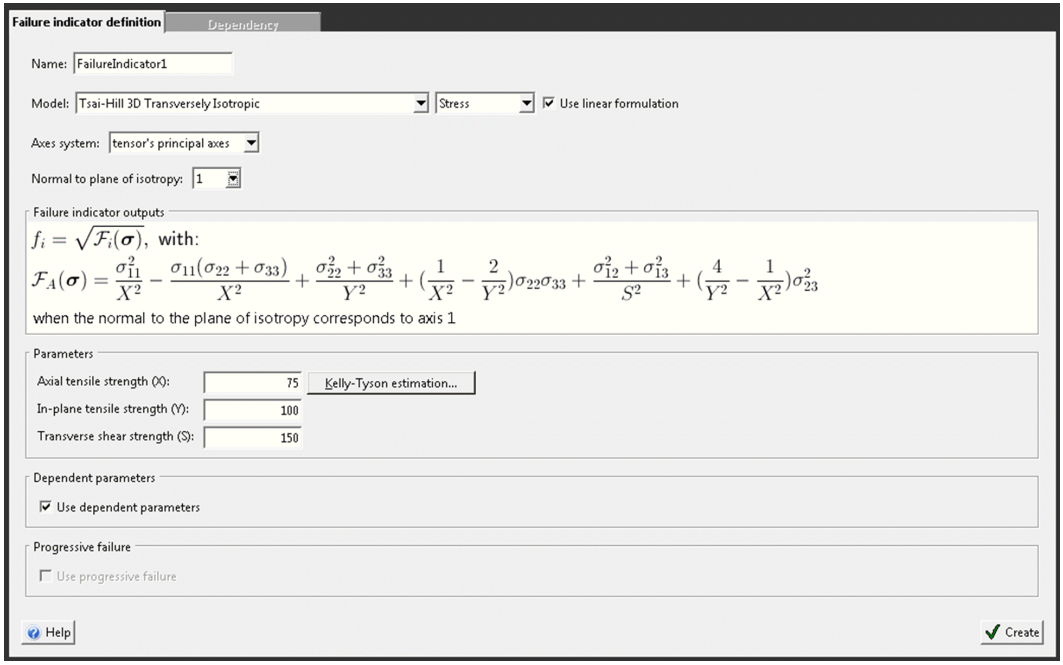


Figure 9-3 Failure indicator definition window.

In the case of 2D local FPGF criteria, the failure criterion is computed in the most constraining local basis.

In the case of 2D local FPGF criteria, the failure criterion is computed in the most constraining local basis. This option is considered by default in the case of the mentioned failure criteria when the inclusion orientation is defined through an orientation tensor. This option may be cancelled

by the user; in that case, the failure criterion is computed in the conventional local basis, that does not correspond necessarily to the most critical one. The choice will be made according to the considered mechanical problem knowing that the local critical basis is a relevant choice in all cases but may induce longer computations. The longer computations are caused by the double evaluation of the stress or strain tensor (once in the conventional basis, and once in the most critical one). This option is called **Use critical basis** and is available in the **Failure** tab under the **First Pseudo-Grain Failure** menu.

Setting Dependencies Over Strength Parameters

When the check box «Use dependent parameters» is checked in the first tab of the failure indicator definition window, a second tab appears that allows the user to define as many dependencies as necessary. All existing dependencies are displayed in a table. Each dependency can be activated or deactivated by clicking on the icon in the first column of the table. Once a dependency is deactivated, it appears grayed in the table and its icon becomes a red cross. All columns in the table of dependencies are editable:

- Strength parameter: define on which parameter the current dependency will apply
- Model: Cowper-Symonds, logarithmic Cowper-Symonds or piecewise linear function
- Variable: Total strain rate, plastic strain rate, temperature, or user-variable
- Parameter: the content of this cell depends on the value selected in the Model cell. It is either the value of the two parameters of the Cowper-Symonds model, or the name of a piecewise linear function.

Remarks:

- The Temperature dependency variable is only available when a temperature loading item is defined, e.g., for a thermo-mechanical analysis.
- Five user-variable are available for dependencies. The user-variable loading item appears when at least of User-variable dependency is activated, and the corresponding failure indicator is assigned and activated.
- User-variable dependencies are not available in Digimat-CAE.

Each dependency can be inspected by clicking on the plot icon in the last column of the table. This opens a new window with a plot of the selected strain rate dependency. Note that by default, the X axis of this plot uses a logarithmic scale for strain-rate dependencies, and a linear scale for other temperature and user-variable dependencies. This scaling can be easily modified trough the plot context menu (right click anywhere in the plot area).

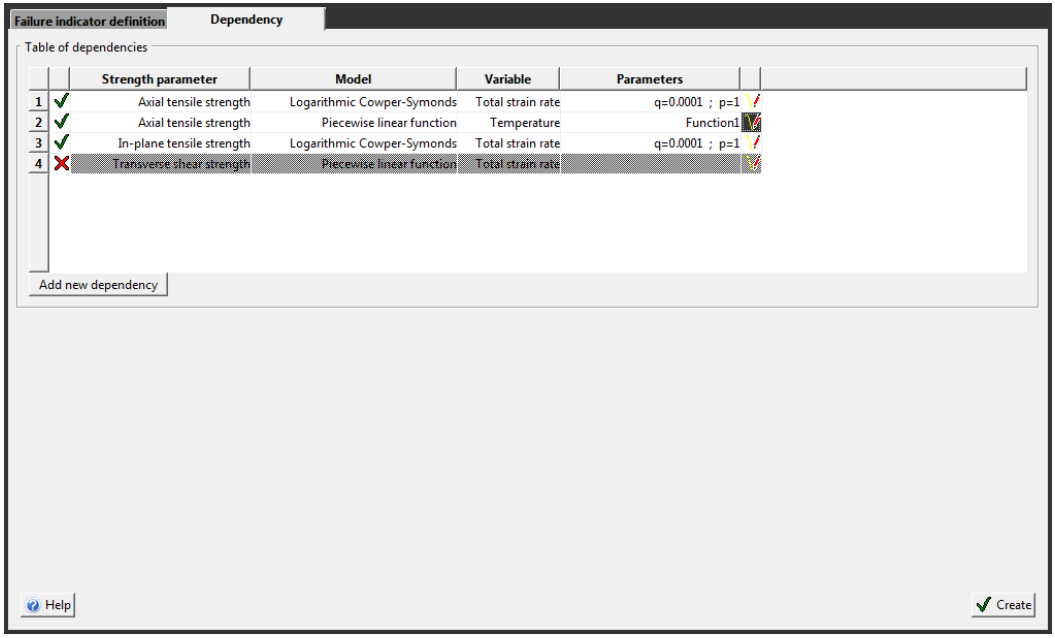


Figure 9-4 Failure dependency window.

Kelly-Tyson Composite Strength Estimator

The Kelly-Tyson formula computes an estimate of a composite's tensile strength based on various matrix/ fiber strength parameters and microstructure parameters. The Kelly-Tyson tensile strength estimate is a composite strength and is thus meant to be used in failure indicators that are applied at the composite level or at the pseudo-grain level.

Kelly and Tyson estimate the composite's tensile strength x^t_1 (subscript 1 indicates that the estimate refers to a tensile strength in the direction of the fiber) using the following formula:

$$x_1^t = \begin{cases} v_f\left(1 - \frac{l_c}{2l}\right)\sigma_f + (1 - v_f)\sigma_m & \text{if } l \geq l_c \\ v_f\left(\frac{\tau l}{D}\right) + (1 - v_f)\sigma_m & \text{if } l < l_c \end{cases} \quad (9\text{-}70)$$

where v_f is the fiber volume fraction, σ_f is the fiber tensile strength, σ_m is the matrix tensile strength, τ is the matrix-fiber interfacial strength, D is the fiber diameter, l is the fiber length, and l_c is a characteristic fiber length given by,

$$l_c = \frac{\sigma_f D}{2\tau}$$

Note that these expressions can be rewritten in terms of fiber aspect ratio AR = l/D, so it is not necessary to know both the fiber length l and the fiber diameter D.

As shown in Figure 9-3, the failure indicator dialog window contains a button labeled Kelly-Tyson estimation. Clicking it opens the Kelly-Tyson strength estimation window. This dialog box allows the user to input the various strength parameters required by the estimator. The microstructure parameters such as the volume fraction of the fiber phase and the aspect ratio of the fibers can directly be extracted from a previously defined microstructure.

Failure Indicator Assignment

Once a failure indicator is defined, it needs to be assigned to the material to be considered by Digimat during the analysis.

The procedure to assign a failure criterion is depicted in Figure 9-5. All failure indicators assignments are defined in the table. New failure indicator assignments can be created by clicking on the **Add assignment** button, or by right clicking in the table area and using the context menu that pops up. Each assignment can be activated or deactivated by clicking on the icon in the first column of the table; an inactive assignment will have its row grayed, and a red cross icon.

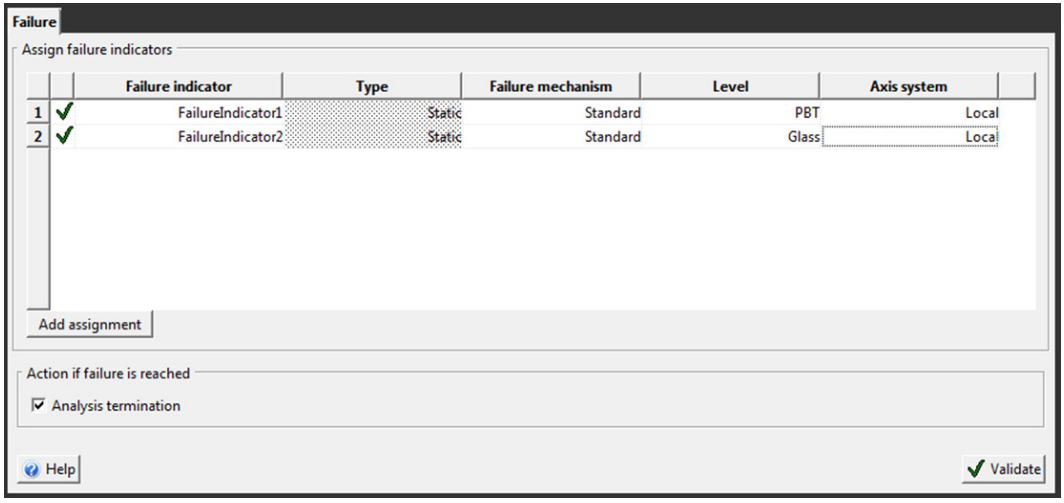


Figure 9-5 Failure criterion assignment.

When adding a Static or Dynamic failure indicator assignment, the user can choose the failure mechanism to use: Standard or FPGF (First Pseudo-Grain Failure – FPGF). More information on the FPGF failure modeling approach can be found in section First Pseudo-Grain Failure Model. These failure indicators can be can be applied at the following levels:

- the composite level; the failure indicators are then computed using the macroscopic stress or strain tensors.
- the material level; the failure indicators are then computed using per phase average stress or strain tensors, in all phases defined with the selected material.

The columns "Type" and "Axis system" are only informative and cannot be edited.

The Fatigue failure indicators activate some specificities, and are automatically applied at the Composite level and in the Local axis system.

The user should also be aware of the limitations reported in section First Pseudo-Grain Failure Model regarding the assignment of FPGF and Fatigue failure indicators.

Element Deletion and Stopping Points

In an explicit analysis, an integration point (RVE) is marked for deletion as soon as:

- if Standard failure is used, any regular failure indicator has reached 1, or
- if FPGF is used, the output of any FPGF criteria has reached 1 (since the output is normalized by the critical value), or
- if Elasto-plasticity with Damage is used, the damage variable reaches the critical damage at which failure is supposed to occur, or
- if Progressive failure is used, the damage comparison variable reaches the critical damage value.

Activation of element deletion is the default in explicit analysis. It can be deactivated by unchecking the appropriate check box in the failure tab.

Note that for explicit analyses, other parameters and flags that are relevant to element deletion need to be set correctly as well. Keep in mind that some explicit codes require all integration points to be marked for deletion before the finite element is actually removed.

The same condition are used in an implicit analysis (including Digimat-MF stand-alone) in order to trigger the possible stop of the analysis. The stop of the analysis in implicit simulation is not activated by default (except for Digimat-MF stand-alone). It can be deactivated by checking (resp. unchecking) the appropriate check box in the failure tab.

Here is an example `*.log` file message for an implicit stop due to failure using FPGF:

```
# DIGIMAT WARNING: Digimat analysis terminated due to failure (FPGF):
```

Failure for Multilayer RVE

When the RVE is defined as a multilayer, some additional options for RVE failure trigger are proposed in the Failure Assignment tab (see Figure 9-6):

- Average failure indicator value: Failure is triggered when the thickness-weighted average value of the failure indicator over all layers reaches the critical value.

Remark: The averaging operation is performed over the maximum value of all failure indicator outputs available for a given layer; when none is a available, a null value is used. Moreover, for the following options, the layer is declared as failed when this maximum value reaches 1.

- Failed thickness fraction: Failure is triggered when the thickness-weighted fraction of failed layers reaches the critical value.
- First-layer failure: Failure is triggered when the first layer fails.
- Specific-layer failure: Failure is triggered when any of the layers indicated in the selection fails.
- All-layer failure: Failure is triggered when all the layers are failed.

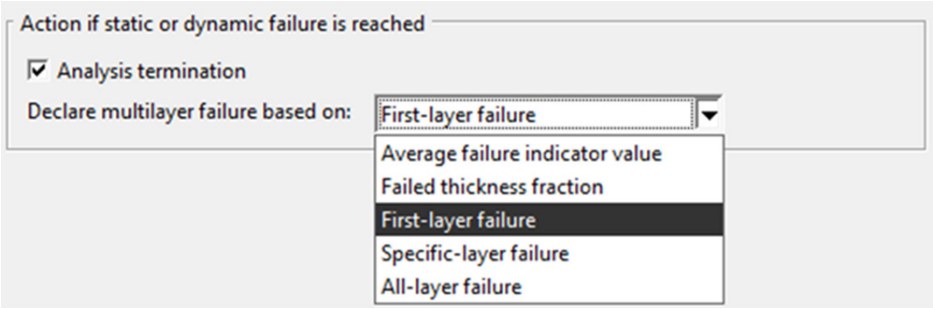


Figure 9-6 Multilayer failure options.

This capability also activates by default some additional outputs at the macroscopic level (see subsection Output for Multilayer Failure), which can be deactivated by unchecking the **MLF** item (under the **Failure item** group) in the **Output definition** tab.

Remark:

- The multilayer failure options only apply to multilayer RVE defined in Digimat-MF. When Digimat is coupled to a Finite Element code, the RVE can only be classical (single-layer); the stacking and failure options are managed on the Finite Element code side.
- The multilayer failure options are not available when a standard failure indicator is assigned at Composite level. In this case, the failure indicator is computed at the macroscopic level (e.g. over the whole RVE), so the notion of layer failure does not apply.
- The multilayer failure options do not apply to fatigue failure indicators, and especially to the pseudo grain fatigue model (see section Fatigue Models).
- When loading a Digimat Analysis file which does not contain the **multilayer failure** keywords, the option selected by default respects the backward compatibility, e.g.: **First-layer** failure when a standard failure indicator is assigned, and **All-layer failure** when a FPGF failure indicator is assigned.

Progressive Failure Model

This section introduces the progressive failure capability of Digimat-MF. The aim of progressive failure is to be able to gradually degrade the mechanical properties of a material during the analysis, in contrast with brittle failure for which analysis is abruptly stopped (or element is deleted, in the context of an explicit finite element analysis). Another interest of progressive failure is to be able to account for anisotropic damage and failure: for instance, a unidirectional ply that fails in the transverse direction can still exhibit a significant stiffness in the fibers direction.

The progressive failure models defined in Digimat mainly target long-fiber uni-directional or orthogonal woven composite materials (laminae or laminates) with an elastic behavior. They use the concept of Continuum Damage Mechanics, which is detailed hereafter. Each progressive failure indicator requires the definition of a failure indicator and a damage model, and can then be assigned to the composite material. Damage variables will be computed all along the analysis, as soon as the failure indicator reaches a given threshold. The elastic properties of the material (e.g. the stiffness) will then be degraded according to these damage variables, until material failure occurs.

Theory

The Continuum Damage Mechanics Framework

Most of the progressive failure models proposed in the literature are developed within the framework of Continuum Damage Mechanics (CDM). Rather than considering initiation and propagation of a single macrocrack in the material, this approach focuses on the influence of micro-defects on the mechanical properties of the material. The application of this framework to composite materials was popularized by Talreja (1985).

For fiber-reinforced unidirectional or woven laminae, the micro-defects can consist in: (see Figure 9-7)

- micro-cracks or micro-voids, that appear in the bulk matrix, and affect mainly the transversal tensile and shear behavior of the material
- fiber debonding, appearing at the fiber-matrix interface, that affects mainly the tensile longitudinal and transversal behavior
- fiber micro-buckling and kinking, that affects mainly the longitudinal compressive behavior
- fiber breakage or pull-out, that affects mainly the longitudinal tensile behavior

Considering a cross-section with normal n within the material, the presence of micro-defects on this surface reduces the area capable of withstanding stress. This implies that each component of the apparent stress vector $T^{(n)} = n \cdot \sigma$, defined as the force divided by the total cross section area, has a value lower than the effective stress vector $\hat{T}^{(n)} = n \cdot \hat{\sigma}$ defined as the force

divided by the defect-free cross section area that withstands the total load. In the CDM framework, this stiffness reduction is taken into account using damage state variables. For a simple uniaxial case (see Figure 9-8), the relation between the effective stress $\hat{\sigma}$ and the apparent stress σ writes as a function of a single damage variable D:

$$\sigma = \underbrace{(1 - D)}_{M} \hat{\sigma} \tag{9-71}$$

The generalization of equation (9-71) yields,

$$\sigma = M(D) : \hat{\sigma} \tag{9-72}$$

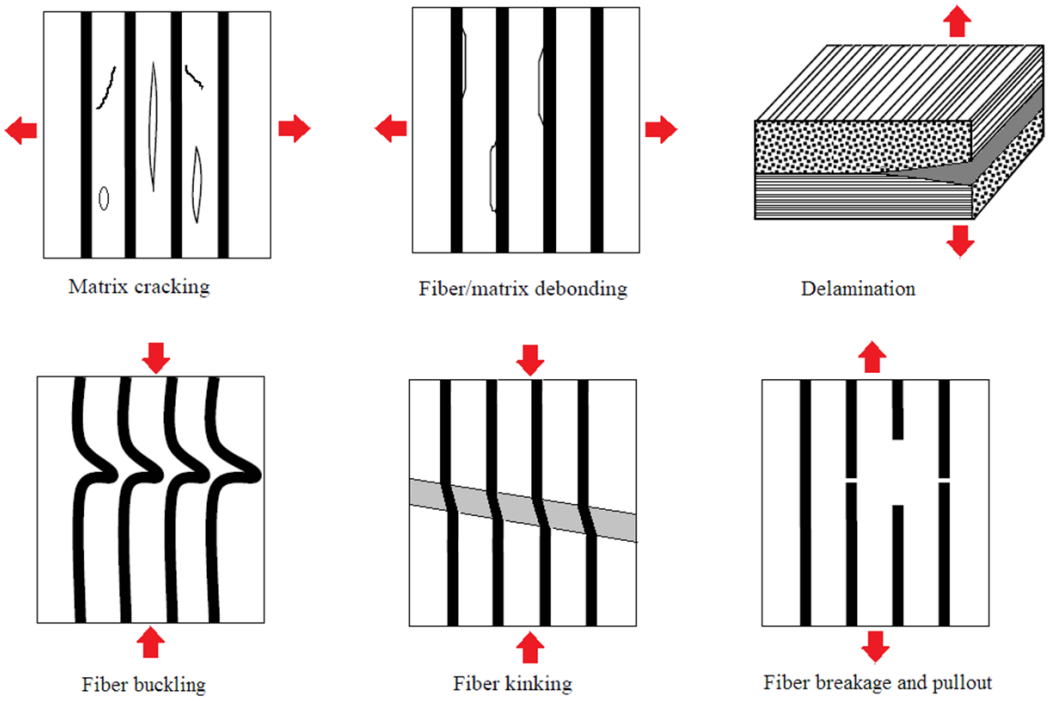


Figure 9-7 Description of typical micro-defects in composite laminae/laminates (adapted from Guangyan (2007)).

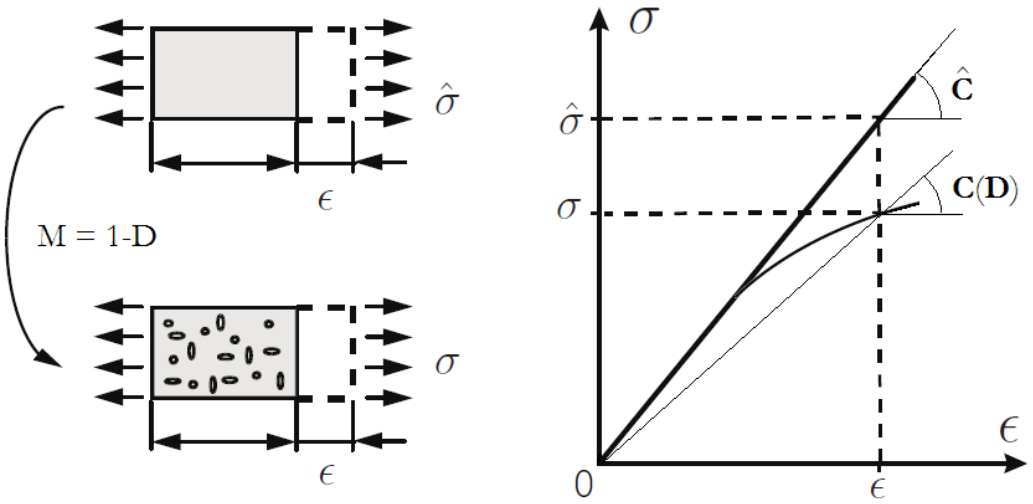


Figure 9-8 Illustration of the concept of effective and damaged stresses (adapted from Guangyan (2007)).

where the damage effect tensor M is a diagonal fourth order tensor depending on the second order damage tensor D and is written in the orthotropy axis of the elastic stiffness as

$$[M] = \begin{bmatrix} 1-D_{11} & 0 & 0 & 0 & 0 & 0 \\ & 1-D_{22} & 0 & 0 & 0 & 0 \\ & & 1-D_{33} & 0 & 0 & 0 \\ & & & 1-D_{12} & 0 & 0 \\ & & & & 1-D_{23} & 0 \\ & & & & & 1-D_{33} \end{bmatrix}$$

The same six components D_{ij} of the damage tensor are used to define the damage compliance S(D), by affecting the diagonal terms of the elastic compliance matrix S in the orthotropy axis of the tensor, i.e.,

$$[S(D)] = \begin{bmatrix} \frac{\hat{S}_{1111}}{1-D_{11}} & \hat{S}_{1122} & \hat{S}_{1133} & 0 & 0 & 0 \\ & \frac{\hat{S}_{2222}}{1-D_{22}} & \hat{S}_{2233} & 0 & 0 & 0 \\ & & \frac{\hat{S}_{3333}}{1-D_{33}} & 0 & 0 & 0 \\ & & & \frac{\hat{S}_{1212}}{1-D_{12}} & 0 & 0 \\ & \text{sym.} & & & \frac{\hat{S}_{2323}}{1-D_{23}} & 0 \\ & & & & & \frac{\hat{S}_{1313}}{1-D_{13}} \end{bmatrix}$$

Considering a material with an initially linear elastic behavior, the relation between the strain and the apparent stress is

$$\sigma = \underbrace{[S(D)]^{-1}}_{C(D)} : \varepsilon,$$

where C(D) is the damaged stiffness tensor.

Along with this formulation, come the definitions of some useful energy densities:

- $E = \frac{1}{2}\sigma : \varepsilon$ is the apparent free energy density
- $\hat{E} = \frac{1}{2}\hat{\sigma} : \hat{\varepsilon}$ is the effective free energy density

  where, $\hat{\varepsilon} = \hat{S} : \hat{\sigma}$
- $E_{acc} = \frac{1}{2}\int(\sigma : d\varepsilon)$ is the accumulated strain energy density

Damage Initiation and Evolution

The initiation of damage in the material is described through the use of a damage evolution indicator, generally expressed as a function of the strain and effective stress tensors:

$$f = f(\varepsilon, \hat{\sigma}) = f_{ini} \text{ at damage initiation} \tag{9-73}$$

This formulation is very similar to that used for classical failure indicators, except that f is expressed from the effective stress tensor, instead of the apparent stress tensor (event though they are equivalent at damage initiation).

The first reason behind the use of effective stresses comes from physical considerations: the micro-defects created when damaging the material do not sustain any loading, and thus should not contribute to the computation of damage. The second reason is that the damage initiation indicator is also used to monitor the evolution of damage; consequently, using the effective stress tensor is mandatory in order to let the damage increase even when the apparent stress decreases.

Remark: In the following, the denomination **failure indicator** will stand for **damage initiation and evolution indicator**, for simplicity.

The evolution of damage is expressed through a damage evolution law, generally in a differential form:

$$\dot{D} = \phi(\dot{f}, f) \tag{9-74}$$

In order to obtain a thermodynamically admissible mechanical behavior, the damage variables range between zero and one, and the damage rate must remain positive or null:

$$\dot{D} \geq 0 \text{ and } 0 \leq D < 1 \tag{9-75}$$

The damage variable is then computed from a function of the failure indicator, the thermodynamical conditions being enforced afterwards:

$$D = \max\varphi(f(\tau)), \text{ with } \varphi \in [0, 1], \tau \in [0, t] \tag{9-76}$$

The damage evolution function φ can take the form of a Heaviside function, a linear function, a power law, a damped exponential function, etc. A list of the damage evolution laws implemented in Digimat is given in subsection Damage Evolution Laws. Moreover, several damage variables and damage evolution laws can be defined in order to represent anisotropic damage of the material; this capability will be discussed hereafter.

Progressive Failure Indicator Definition & Assignment

This subsection introduces the procedure to define and assign a progressive failure indicator in Digimat-MF. It also explains the main limitations to the progressive failure capabilities of Digimat. The procedure to define and assign a progressive failure model consists in two steps:

1. Definition of a progressive failure indicator, attached to the failure indicator
2. Assignment of the progressive failure indicator to a material.

Progressive Failure Indicator Definition

The definition of a progressive failure indicator requires the definition of:

- a failure indicator, which gives the basis on which the damage variables will be computed

- a damage model, which determines which damage variables will be affected, and how
- a damage evolution law, which determines the evolution of the damage variables with the failure indicator

The list of available combinations is given in subsection Damage Models and subsection Damage Evolution Laws.

A progressive failure indicator can be added to the analysis by right-clicking on the failure item in the Digimat tree, and selecting **Add progressive failure indicator** in the context menu that pops up. This action with create and assign a progressive failure indicator with properly defined default parameters. It is also possible to transform a classical failure indicator into a progressive failure indicator, by activating the **Use progressive failure** checkbox in the Failure indicator tab (see Figure 9-9).

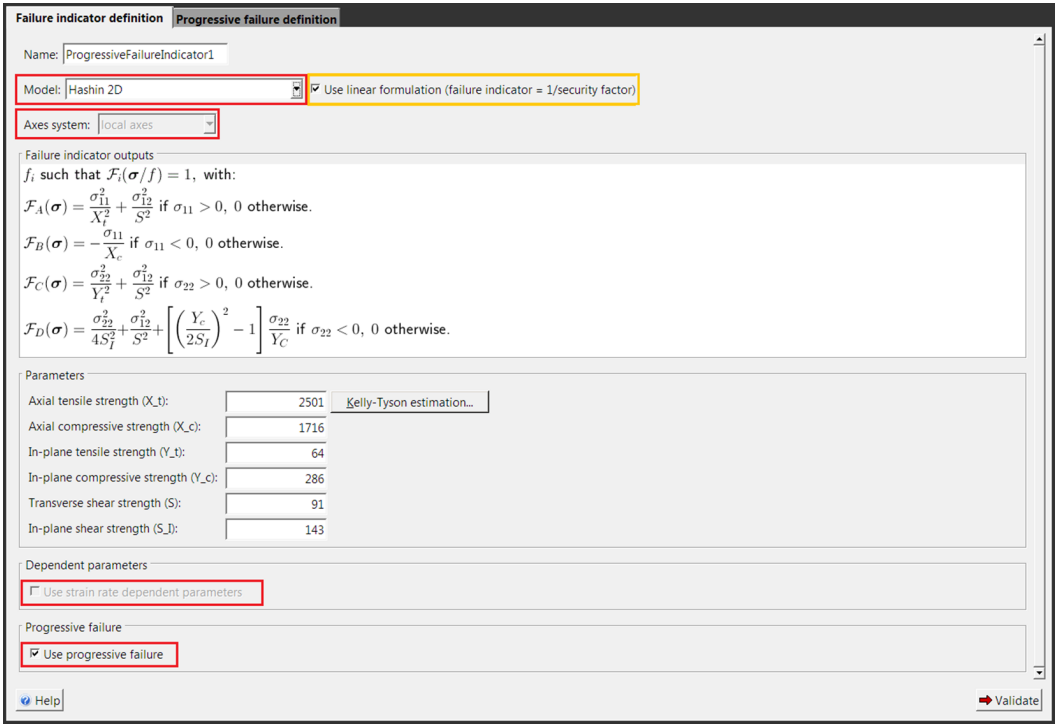


Figure 9-9 Definition of failure parameters for progressive failure.

However, this option is only available when some conditions are met:

- the failure indicator must be a one of the following: Multi-components 2D model, Hashin-class model (Hashin 2D, Hashin-Rotem 2D or Hashin 3D), or User-defined model
- the **Axes system** must be set to **local axes**

- and the **Use dependent parameters** checkbox must be unset.

When done so, a new tab **Progressive failure definition** becomes available, where the user will be able to define the damage parameters associated to this progressive failure indicator. The damage model will be automatically set from the failure indicator model. The user can then choose which damage law will be used, and the associated parameters (see subsection Damage Evolution Laws for a comprehensive description of the damage laws available in Digimat). Typical **Stress vs. Strain** and **Damage vs. Failure indicator** curves are also displayed in the tab, and provide some guidelines for the effect of the damage evolution law and parameters.

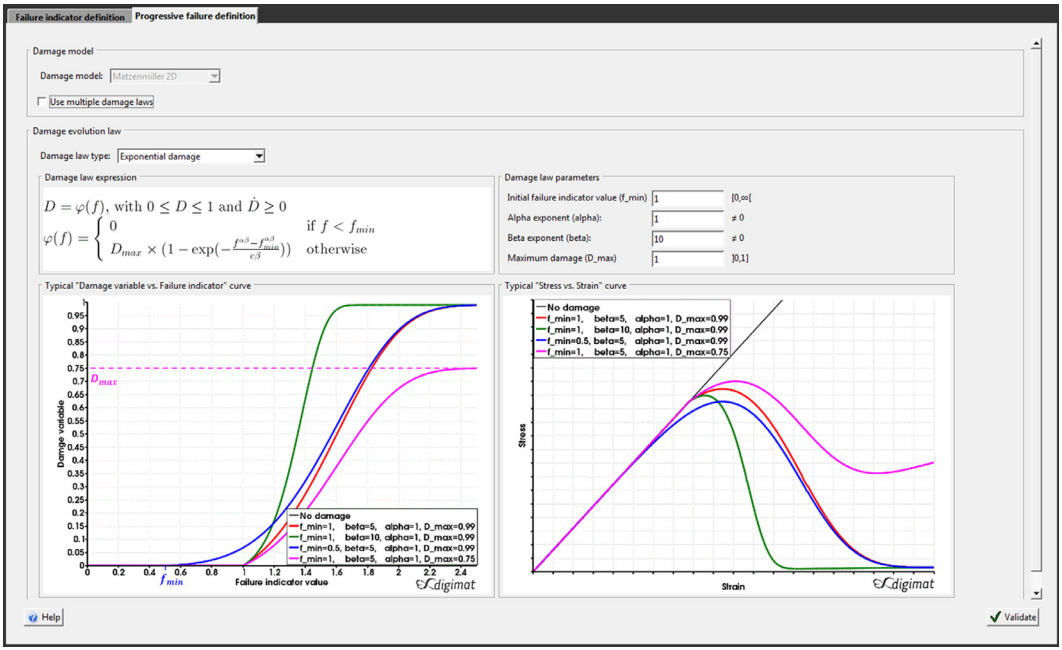


Figure 9-10 Definition of a unique damage evolution law for progressive failure.

Since Digimat 6.0.1, it is also possible to define several damage evolution laws by checking the **Use multiple damage laws** checkbox. This will display a table with a list of individual relations between the failure indicator outputs and the damage variables (see Figure 9-11). The items of this list are specific to each progressive failure model (see subsection Damage Models). This enables to trigger, for example, an instantaneous damage evolution for tension in the fiber's direction, and a much slower damage evolution for tension in the transverse direction.

Remark: This new capability is retro-compatible with previous input format: the **unique** damage law (old input format) is used as an initialization default for the **multiple** damage laws.

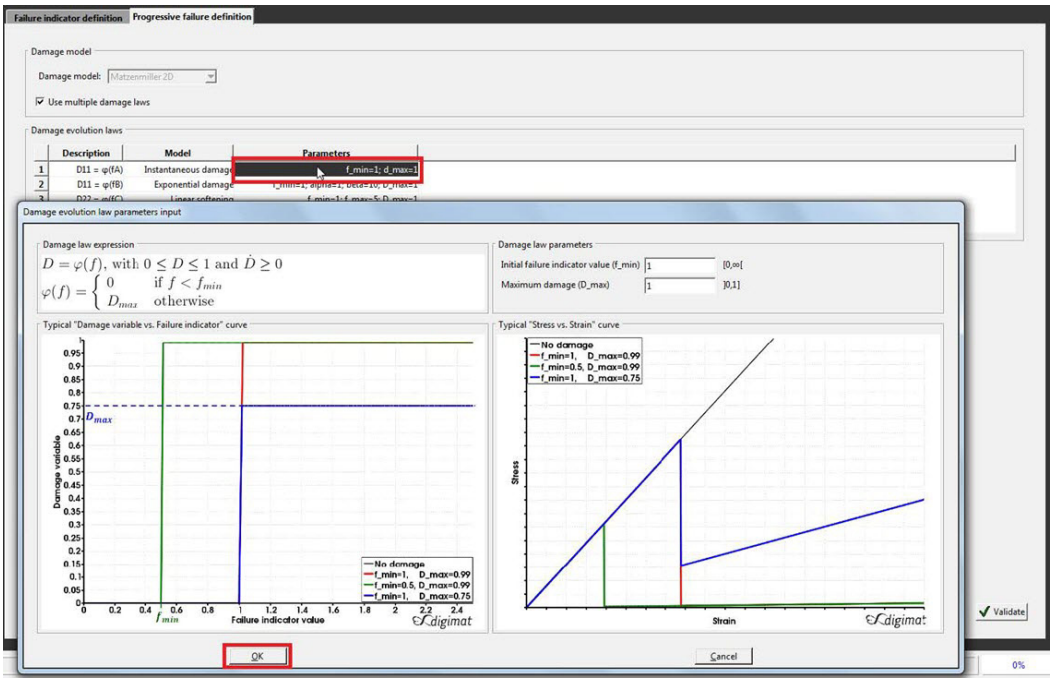


Figure 9-11 Definition of multiple damage evolution laws for progressive failure.

Progressive Failure Indicator Assignment & Controls

After a progressive failure indicator is defined, it must be assigned to a material to be considered by Digimat during the analysis. The procedure is very similar to the assignation of a Standard failure indicator (see subsection Failure Indicator Assignment). When assigning the Progressive Failure indicator, it will appear in the failure indicator assignment table, with the following settings:

- the **Type** is automatically set as **Progressive failure**;
- the **Failure mechanism** is automatically set as **Standard** (as the failure indicator is evaluated from per-phase mechanical states);
- the **Level** is automatically set as **Composite** (only macroscopic damage models are available for now);
- the **Axis system** is automatically set as **Local**.

Several progressive failure controls can also be applied to the analysis.

- The Maximum damage parameter enables to threshold all the damage variables applied at any material level. This parameter enables to limit the damage without changing the damage evolution law parameters (which affect the stress-strain curve aspect). In order to avoid convergence issues, it is recommended to systematically activate the **Maximum damage** parameter with a value slightly below 1 (typically 0.999).

- The Critical damage parameter can be used to trigger analysis stop in Digimat-MF, or element deletion in an explicit coupled simulation (see subsection Element Deletion and Stopping Points).
  - The Maximum damage variable option triggers analysis stop when at least one damage variable (out of the six available) reaches the critical damage value:

$$\max(D_i)_{i \in \{11,\ 22,\ 33,\ 12,\ 23,\ 13\}} \geq D_{crit} \tag{9-77}$$

This option is the less conservative, as it will only require the material to be critically damaged in one given direction to trigger analysis stop.

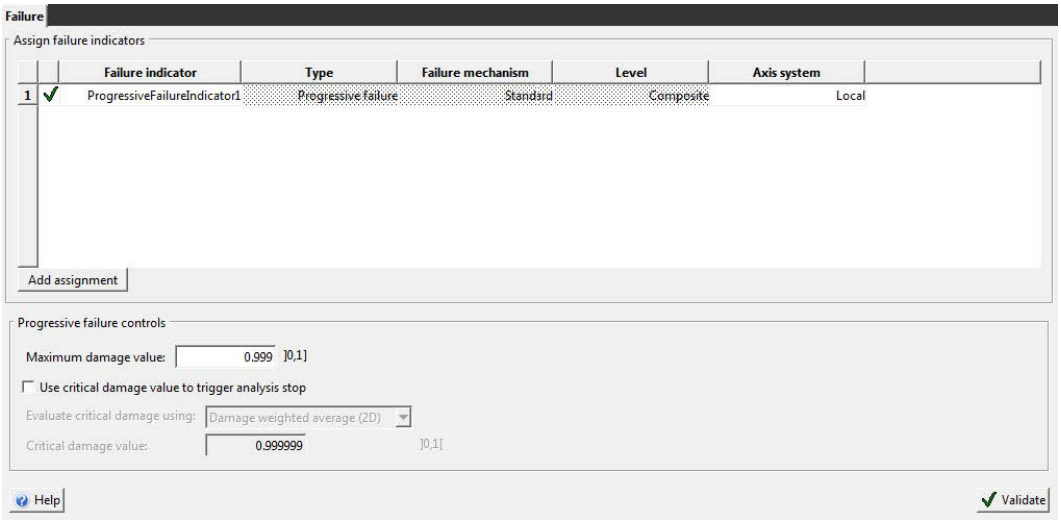


Figure 9-12 Progressive failure indicator assignment.

  - The All damage variables option triggers analysis stop when all six damage variables reach the critical damage value:

$$D_i \geq D_{crit} \quad \text{for all } i \in \{11,\ 22,\ 33,\ 12,\ 23,\ 13\} \tag{9-78}$$

This option is the most conservative, as it requires the material to be critically damaged in all directions to trigger analysis stop. Moreover, some progressive failure models (such as the Matzenmiller 2D model) will never be able to trigger analysis stop with this option.

  - The Free energy dissipation option triggers analysis stop when the free energy dissipation reaches the critical value:

$$1 - \frac{E}{\hat{E}} = 1 - \frac{\frac{1}{2}\sigma : \varepsilon}{\frac{1}{2}\hat{\sigma} : \hat{\varepsilon}} \geq D_{crit} \tag{9-79}$$

This option is the default and recommended setting.

- The Damage weighted by 2D moduli option triggers analysis stop when the following evaluator reaches the critical damage value:

$$D_{weighted2D} = \frac{D_{11}E_1 + D_{22}E_2 + D_{12}G_{12}}{E_1 + E_2 + G_{12}} \geq D_{crit} \quad (9\text{-}80)$$

where E_1, E_2 and G_{12} stand for the engineering moduli of the composite in its orthotropy axis system.

- The Damage weighted by 3D moduli option triggers analysis stop when the following evaluator reaches the critical damage value:

$$D_{weighted3D} = \frac{D_{11}E_1 + D_{22}E_2 + D_{33}E_3 + D_{12}G_{12} + D_{13}G_{13} + D_{23}G_{23}}{E_1 + E_2 + E_3 + G_{12} + G_{13} + G_{23}} \geq D_{crit} \quad (9\text{-}81)$$

Damage Time Integration

In order to ensure convergence of Digimat-MF and Digimat-CAE analyses, damage is computed from the material state at the end of the previous step (i.e., at the beginning of the current step). However, this method induces a significant timestep sensitivity. Consequently, it is recommended to use it in conjunction with a small time step, in order to reduce the delay effect and to obtain smooth stress-strain curves.

Remark: All damage time integration controls available in versions prior to 2018.0 are deprecated.

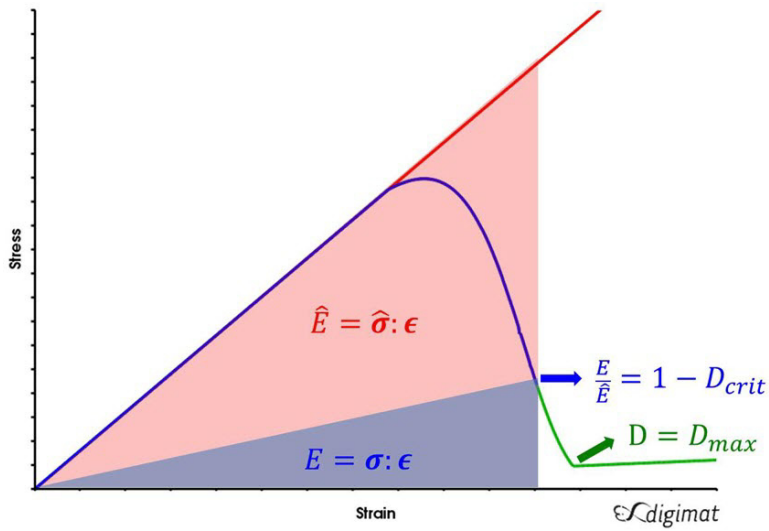


Figure 9-13 Computation and application of progressive failure controls.

Combination of Failure/Progressive Failure Indicators

Several progressive failure indicators can be assigned to the same material. In this case, their damage variables will be combined following the relation:

$$1 - D^{overall} = \prod_{i=1}^{n} (1 - D^{i}) \text{ with i being the progressive failure indicator index} \quad (9\text{-}82)$$

It is also possible to assign simultaneously, for the same material, a Progressive Failure indicator and a Standard failure indicator. If the corresponding indicators are stress-based, the Standard failure indicator will be computed from the apparent stress tensor, whereas the Progressive Failure indicator will be computed from the effective stress tensor. However, analysis stop (or element deletion for Digimat-CAE) will not be triggered if the progressive failure indicator reaches the critical value.

Model-related Limitations

Due to theoretical and technical reasons, the progressive failure capability of Digimat suffers several limitations, which are summed up in subsection Progressive Failure Model. The main limitations are the following:

1. Currently, only macroscopic progressive failure is implemented in Digimat. As a side consequence, the FPGF scheme is not applicable with progressive failure.
2. Progressive failure indicators can only be assigned to materials with a linear elastic behaviour, without dependencies, and without thermal expansion effects. The Continuum damage Mechanics framework, as detailed in subsection Theory, is based on a linear elastic behaviour of the undamaged material (although some declinations for elastoplastic behaviour can be found in the literature).
3. Progressive failure can only be applied into a local axis system. The description of damage with only six damage variables makes it applicable only to an orthotropic stiffness matrix. For this reason, the failure indicator and damage variables must be computed into the axes of orthotropy of the stiffness matrix. This definition also makes sense from a physical point of view, as some models (such as the Matzenmiller model) must be used is the fibers' axis system. For the same reason, it is not available with non-orthotropic woven microstructures.
4. Progressive failure is not applicable to a multilayer RVE in Digimat-MF. Progressive failure is based on a Standard (per-phase) evaluation of the failure indicator, which cannot be evaluated at the ply level (currently). A workaround to this issue is to define a coupled Digimat-CAE analysis on a single shell element with several layers, as the multi-layer capability is then managed by the Finite Element code.

Damage Models

This subsection intends to describe the progressive failure (damage) models implemented in Digimat. All those models are mainly used in the field of long fibers composites. The progressive failure models currently available in Digimat are the following:

- General Anisotropic damage
- Multi-components 2D damage
- Matzenmiller 2D damage
- Matzenmiller 3D damage
- Camanho Damage

General Anisotropic Damage

This model is a generic 3D anisotropic damage model, which is meant to be used with a **user-defined** failure indicator. The user-defined failure indicator implementation must output at least 6 failure indicators, which are used to evaluate the 6 corresponding anisotropic damage variables, through the following equation; if the failure indicator outputs more than 6 values, these additional values are not considered for damage computation.

$$\begin{aligned} D_{11} &= \varphi_A(f_A) \\ D_{22} &= \varphi_B(f_B) \\ D_{33} &= \varphi_C(f_C) \\ D_{12} &= \varphi_D(f_D) \\ D_{23} &= \varphi_E(f_E) \\ D_{13} &= \varphi_F(f_F) \end{aligned} \tag{9-83}$$

The association between failure mechanisms and anisotropic damage variables must be managed by the implementation of the user-defined failure indicator. Consequently, it is not possible to specify combinative relations between damage variables.

Remark: For a more comprehensive definition of damage evolution, it is recommended to define the failure indicator such that its outputs evolve linearly with stress and strain (see subsection General Definition).

Multi-components 2D Damage

The multi-components 2D model is a simple damage model to be applied to long-fibers composite materials. It requires the definition of a multi-components 2D failure indicator, and the damage variables are computed as:

1. longitudinal damage (fiber-related, with traction-compression difference):

$$D_{11} = \varphi_A(f_A) \text{ if } f_A > 0;$$
$$D_{11} = \varphi_B(f_B) \text{ otherwise} \tag{9-84}$$

2. transversal damage (matrix-related, with traction-compression difference):

$$D_{22} = \varphi_C(f_C) \text{ if } f_C > 0;$$
$$D_{22} = \varphi_D(f_D) \text{ otherwise} \tag{9-85}$$

3. in-plane shear damage:

$$D_{12} = \varphi_E(f_E) \tag{9-86}$$

4. all the other damage variables are null:

$$D_{33} = D_{13} = D_{23} = 0 \tag{9-87}$$

Remark: This model is a simplification of the Matzenmiller 2D model presented hereafter.

Matzenmiller 2D Damage

The Matzenmiller-Lubliner-Taylor (MLT) model, or Hashin damage model (Matzenmiller et al. (1995)), is mainly applied to long-fibers composite materials. It requires the definition of Hashin-class 2D failure indicators (Hashin 2D or Hashin-Rotem 2D), and the damage variables are computed as:

1. longitudinal damage (fiber-related, with traction-compression difference):

$$D_{11} = \varphi_A(f_A) \text{ if } f_A > 0;$$
$$D_{11} = \varphi_B(f_B) \text{ otherwise} \tag{9-88}$$

2. transversal damage (matrix-related, with traction-compression difference):

$$D_{22} = \varphi_C(f_C) \text{ if } f_C > 0;$$
$$D_{22} = \varphi_D(f_D) \text{ otherwise} \tag{9-89}$$

3. in-plane shear damage (combinative):

$$D_{12} = 1 - (1 - D_{11})(1 - D_{22}) \tag{9-90}$$

4. all the other damage variables are null:

$$D_{33} = D_{13} = D_{23} = 0 \tag{9-91}$$

Matzenmiller 3D Damage

This model is a 3D extension of the original MLT model, and is also applied to long-fibers composites. It requires the definition of a Hashin 3D failure indicator, and the damage variables are computed as:

1. longitudinal damage (fiber-related, with traction-compression difference):

$$D_{11} = \varphi_A(f_A) \text{ if } f_A > 0;$$
$$D_{11} = \varphi_B(f_B) \text{ otherwise} \tag{9-92}$$

2. transversal damage (matrix-related, with traction-compression difference):

$$D_{22} = D_{33} = \varphi_C(f_C) \text{ if } f_C > 0;$$
$$D_{22} = D_{33} = \varphi_D(f_D) \text{ otherwise} \tag{9-93}$$

3. longitudinal shear damage (combinative):

$$D_{12} = D_{13} = 1 - (1 - D_{11})(1 - D_{22}) \tag{9-94}$$

4. transversal shear damage (combinative):

$$D_{23} = D_{22} \tag{9-95}$$

Camanho Damage

The Camanho model is designed to be applied to unidirectional composite materials and can only be used in Digimat in conjunction with the Camanho failure indicator. All the details of the model can be found in the paper by Maimí et al. (2007).

The model makes use of five intrinsic damage variables: D_{11}^T, D_{11}^C, D_{22}^T, D_{22}^C and D_{SL}. These keep track of the degradation of the in-plane material behavior under the five uniaxial loading conditions below:

- tensile loading in the longitudinal direction: D_{11}^T
- compression loading in the longitudinal direction: D_{11}^C
- tensile loading in the transverse direction: D_{22}^T
- compression loading in the transverse direction: D_{22}^C
- in-plane shear loading: D_{SL}

The six classical damage variables controlling the behavior of the composite material are computed from these intrinsic damage variables as a function of the actual stress-state being seen by the material:

1. longitudinal damage (with traction-compression difference):

$$\begin{aligned} D_{11} &= D_{11}^{T} \text{ if } \varepsilon_{11} > -(1 - D_{22}^{C})\nu_{21}\varepsilon_{22}; \\ D_{11} &= D_{11}^{C} \text{ if } \varepsilon_{11} < -(1 - D_{22}^{T})\nu_{21}\varepsilon_{22}; \\ D_{11} &= \frac{1}{2}(D_{11}^{T} + D_{11}^{C}) \text{ otherwise} \end{aligned} \tag{9-96}$$

2. transversal damage (with traction-compression difference):

$$\begin{aligned} D_{22} &= D_{22}^{T} \text{ if } \varepsilon_{22} > -(1 - D_{11})\nu_{12}\varepsilon_{11}; \\ D_{22} &= D_{22}^{C} \text{ otherwise} \end{aligned} \tag{9-97}$$

3. out-of-plane damage (combinative):

$$D_{33} = 1 - (1 - D_{22}^{C})(1 - D_{11}^{C}) \tag{9-98}$$

4. In-plane shear damage (combinative):

$$D_{12} = 1 - (1 - D_{SL})(1 - D_{11}^{T}) \tag{9-99}$$

5. longitudinal shear damage:

$$D_{13} = D_{11}^{T} \tag{9-100}$$

6. transversal shear damage:

$$D_{23} = D_{SL} \tag{9-101}$$

The values of the five intrinsic damage variables are computed from the outputs of the Camanho failure indicator as follows:

$$\begin{aligned} D_{11}^{T} &= D_{11}^{T}(\max\{f_A, f_B\}) \\ D_{11}^{C} &= D_{11}^{C}(f_B) \\ D_{22}^{T} &= D_{22}^{T}(\max\{f_C, f_D\}) \\ D_{22}^{C} &= D_{22}^{C}(f_D) \\ D_{SL} &= D_{SL}(\max\{f_C, f_D\}) \end{aligned} \tag{9-102}$$

It can be seen in the above equations that the intrinsic damage variables keeping track of the material degradation under tensile and shear loads – D_{11}^{T}, D_{22}^{T}, and D_{SL} – depend upon the failure indicators for either both longitudinal failure modes or both transverse failure modes. This models the fact that the behavior of the material under tensile and shear loads is affected by the

two types of failure modes. The reason behind is that all cracks tend to open up under tensile and shear loads regardless of the failure mode which they are resulting from. Conversely, the intrinsic damage variables keeping track of the material degradation under compressive loads – D_{11}^{C}, and D_{22}^{C} – depend upon a single failure indicator. This models the fact the behavior of the composite material under longitudinal or transverse compressive loads is only affected by longitudinal failure under compression loads or transverse failure under compression-dominated loads. The reason behind is that the cracks resulting from tensile failure modes tend to close under compression loads so that the material recovers some or all of its load carrying capability.

The actual damage evolution laws used in the above equations to compute the intrinsic damage variables are such that the stress-strain behavior exhibits:

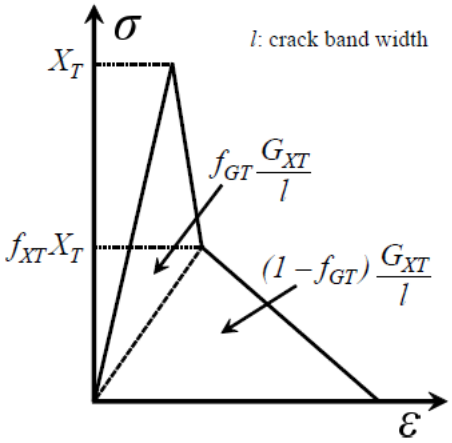


Figure 9-14 Stress-strain behavior in longitudinal traction.

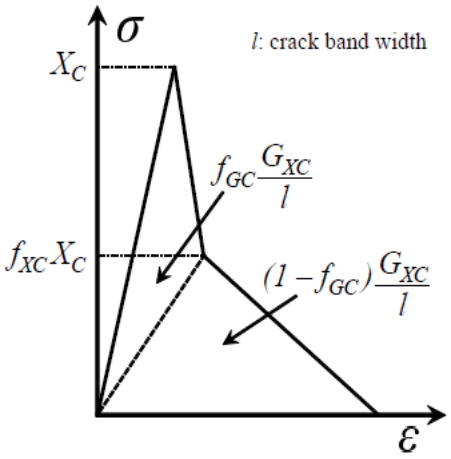


Figure 9-15 Stress-strain behavior in longitudinal compression.

- bilinear softening under monotonic tensile loading in the longitudinal direction (see Figure 9-14)
- bilinear softening under monotonic compression loading in the longitudinal direction (see Figure 9-15)
- linear softening under monotonic tensile loading in the transverse direction (see Figure 9-16)
- linear softening under monotonic compression loading in the transverse direction (see Figure 9-17)
- linear softening under monotonic in-plane shear loading (see Figure 9-18)

The different parameters controlling the softening behavior in Figure 9-14–Figure 9-18 are listed below:

- Longitudinal tension strength, $X_T > 0$.
- Longitudinal tension strength ratio at inflection point, $0 < f_{XT} < 1$.
- Fracture toughness in longitudinal tension, $G_{XT} > 0$.
- Proportion of G_{XT} dissipated by the first part, $0 < f_{GT} < 1$.
- Longitudinal compression strength, $X_C > 0$.
- Longitudinal compression strength ratio at inflection point, $0 < f_{XC} < 1$.
- Fracture toughness in longitudinal compression, $G_{XC} > 0$.
- Proportion of G_{XC} dissipated by the first part, $0 < f_{GC} < 1$.
- Transverse tension strength, $Y_T > 0$.

- Fracture toughness in transverse tension, $G_{YT} > 0$.
- Transverse compression strength, $Y_C > 0$.
- Fracture toughness in transverse compression, $G_{YC} > 0$.
- In-plane shear strength, $S_L > 0$.
- Fracture toughness in in-plane shear, $G_{SL} > 0$.
- Crack bandwidth l

When using a progressive failure model, the cracks which appear in the material when it fails are not explicitly modeled through material discontinuities. Their presence is rather taken into account through the use over a certain distance across the actual crack of a modified stress-strain behavior exhibiting e.g., some softening. The crack bandwidth l appearing in the above list of parameters is precisely that finite distance. When using the Camanho model described above in a finite element analysis, l must therefore be taken equal to the size of the finite elements in the direction normal to the crack plane. Most frequently, it is simply taken equal to the characteristic length of the elements returned by the host finite element code. Please note, however that the definition of that characteristic length of the elements might slightly differ from one finite element code to the other. When using this progressive failure model through Digimat-CAE, this crack bandwidth parameter must be set equal to the thickness of a single ply in the laminate.

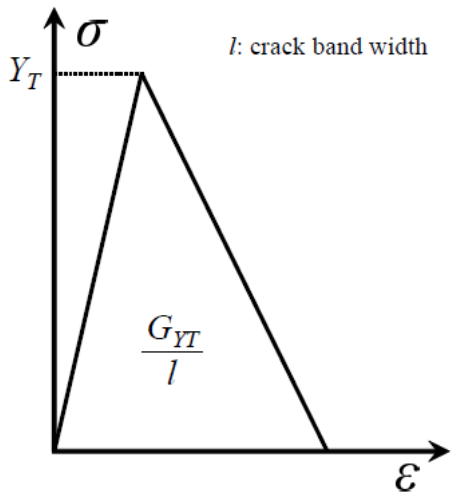


Figure 9-16 Stress-strain behavior in transversal traction.

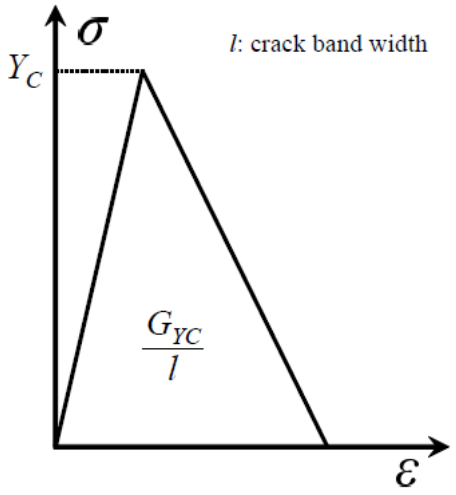


Figure 9-17 Stress-strain behavior in transversal compression

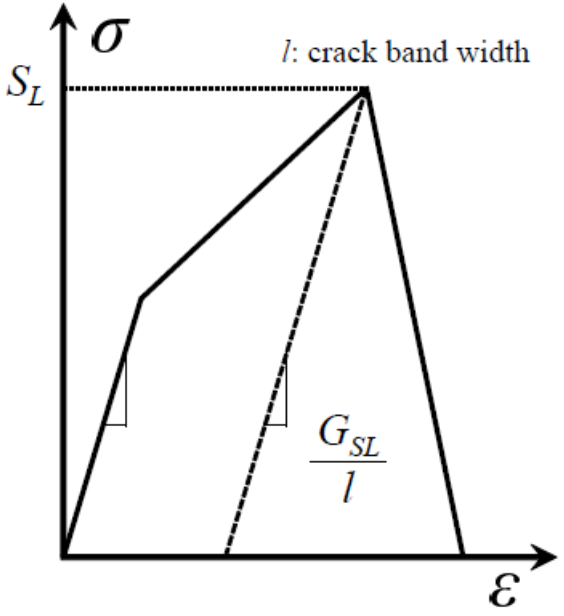


Figure 9-18 Stress-strain behavior in shear.

Damage Evolution Laws

This subsection intends to describe the damage evolution laws that are implemented in Digimat. Currently, the following damage evolution laws are available:

- No-damage evolution law
- Instantaneous damage evolution law
- Power damage evolution law
- Exponential damage evolution law
- Linear softening damage evolution law

All the damage variables are evaluated with the same evolution law (although from different failure indicators), and with some additional consistency relations:

$0 \leq \mathrm{D}(f) < 1$ (positive threshold damage), and $\dot{\mathrm{D}}(f) \geq 0$ (irreversible damage) (9-103)

No-damage Evolution Law

This evolution law requires no parameter, and outputs a null damage value:

$$\varphi(f) = 0 \tag{9-104}$$

This damage law is mainly useful for testing duties, or to perform a non-intrusive evaluation of a progressive failure indicator (i.e., a failure indicator evaluated from effective stresses), when another progressive failure indicator is already assigned to the material.

Instantaneous Damage Evolution Law

This evolution law requires two parameters $f_{\min}$ and $\mathrm{D}_{\max}$, and triggers an instantaneous evolution of damage:

$$\varphi(f) = \begin{cases} 0 & \text{if } f < f_{\min} \\ \mathrm{D}_{\max} & \text{otherwise} \end{cases} \tag{9-105}$$

This damage law roughly corresponds to a brittle failure mechanism, but with an anisotropic effect. For an uniaxial loading, the stress-strain curve typically shows a brutal decrease when the failure indicator value reaches $f_{\min}$, then continues with a lower slope (1 - $\mathrm{D}_{\max}$)E.

Typical parameters values for this evolution law are:

- $f_{\min} = 1$ and $\mathrm{D}_{\max} = 1$ (complete damage).

Power Damage Evolution Law

This evolution law requires five parameters $f_{\min}$, $f_{\max}$, α, $\mathrm{D}_{\max}$ and $\mathrm{D}_{\text{final}}$, and triggers a power-law evolution of damage w.r.t. the failure indicator value:

$$\varphi(f) = \begin{cases} 0 & \text{if } f < f_{min} \\ D_{max} \dfrac{f^{\alpha} - f_{min}^{\alpha}}{f_{max}^{\alpha} - f_{min}^{\alpha}} & \text{if } f_{min} \le f < f_{max} \\ D_{final} & \text{otherwise} \end{cases} \quad (9\text{-}106)$$

Typical parameters values for this evolution law are:

- $f_{min} = 1$, $\alpha = 1$, $D_{max} = D_{final} = 1$ and a variable value for f_{max} (linear damage, see subsection Progressive Failure Model).
- $f_{min} = 0$, $D_{max} = D_{final} = 1$ and variable values for f_{max} and α (power-law damage).

Remark: The D_{final} parameter enables to trigger an instantaneous damage evolution after a given value of the value indicator. However, in most cases, one can simply set $D_{max} = D_{final}$.

Exponential Damage Evolution Law

This evolution law requires four parameters f_{min}, and D_{max}, and triggers a damped exponential evolution of damage w.r.t. the failure indicator value:

$$\varphi(f) = \begin{cases} 0 & \text{if } f < f_{min} \\ D_{max}\left(1 - \exp\left(-\dfrac{f^{\alpha\beta} - f_{min}^{\alpha\beta}}{e\beta}\right)\right) & \text{otherwise} \end{cases} \quad (9\text{-}107)$$

Typical parameters values for this evolution law are:

- $f_{min} = 1$, $\alpha = 1$, $D_{max} = 1$ and a variable value for β.
- $f_{min} = 0$, $\alpha = 1$, $D_{max} = 1$ and a variable value for β, which results in a Weibull-shaped stress-strain curve, as discussed in subsection Progressive Failure Model).

Linear Softening Damage Evolution Law

This evolution law requires four parameters f_{min}, f_{max}, D_{max} and D_{final}, and triggers the following evolution of damage w.r.t. the failure indicator value:

$$\varphi(f) = \begin{cases} 0 & \text{if } f < f_{min} \\ D_{max} \dfrac{f_{max}}{f} \dfrac{f - f_{min}}{f_{max} - f_{min}} & \text{if } f_{min} \le f < f_{max} \\ D_{final} & \text{otherwise} \end{cases} \quad (9\text{-}108)$$

Remark: An equivalent expression can be obtained by using a Power damage evolution law with α = -1.

Typical parameters values for this evolution law are:

- f_{min} = 1, D_{max} = 1 and a variable value for f_{max}, which results in a bilinear-shaped stress-strain curve.

Remark: The D_{final} parameter enables to trigger an instantaneous damage evolution after a given value of the value indicator. However, in most cases, one can simply set $D_{max} = D_{final}$.

First Pseudo-Grain Failure Model

Digimat proposes two failure mechanisms. There is the standard approach for which, by verifying whether the assigned failure criterion is reached, you control if the RVE breaks or not. This is a sort of binary approach because there are only two possibilities: sane RVE or broken RVE. Since version 3.1, Digimat has introduced a new progressive failure mechanism that is called the First Pseudo-Grain Failure (FPGF) model.

This approach is different in the sense that the RVE failure is progressive, as it will be explained in this section of the documentation, such that the stiffness of the RVE progressively decreases as is the case when modeling damage. This model is developed specifically for short fibers reinforced polymer composites.

This page has for objective to explain in details how the FPGF model works.

Theory

FPGF model - Concept of Pseudo-grain

Figure 9-19 introduces the concept of pseudo-grain for composites reinforced with short fibers inclusions in a RVE, which is the core of the FPGF model:

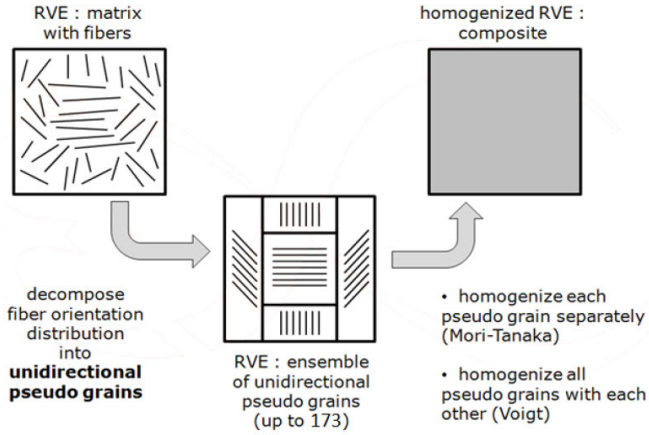


Figure 9-19 The orientation distribution of the fibers (top left) is discretized into a limited number of aligned 'grains', called pseudo-grains (bottom). The pseudo-grains are homogenized in two levels (first individually, then collectively) to obtain the homogeneous composite (top right). RVE stands for Representative Volume Element.

In a real composite reinforced with fibers, the fibers are described by an orientation distribution (often represented by an orientation tensor). The basis of the FPGF model is to discretize the orientation distribution (Figure 9-19, top left) into a limited number of perfectly aligned 'grains', called pseudo-grains (bottom of Figure 9-19). One single pseudo-grain can thus be considered as an internal, aligned composite, containing both the matrix phase and the fiber phase. Each pseudo-grain is a strictly 2-phases composite, a matrix combined with an inclusion phase. Note that pseudo-grains are a concept, a numerical artifact to perform mean-field homogenization on RVE presenting a non-fixed inclusion orientation. The pseudo-grains cannot be seen in the real composite.

As stated previously, the idea behind the FPGF model is to decompose the orientation distribution, expressed by a tensor, into a limited number of grains, each containing a perfectly aligned orientation of the inclusions.

But one might ask: **How is the orientation distribution exactly decomposed?** The explanation is intrinsically linked to the angle increments definition. The angular space can be illustrated by a sphere. This sphere is divided into many segments of equal angular length, with horizontal and vertical lines, as it is done for the planet Earth with the meridians and the parallels. From their position in the sphere of angular space, all segments describe one unique individual zone of the angular space.

Getting back to the FPGF model, what must be understood is that each pseudo-grain is equivalent to one angular segment, meaning it expresses one unique area of the angular space. So every pseudo-grain, in which inclusions are modeled as perfectly aligned, expresses a different state of orientation.

Unless the inclusions are oriented in a random 3D state, in which case the importance of all pseudo-grains is the same, some orientation states are always more important than some others.

To account for that, a weight is assigned to each pseudo-grain. If the inclusions tend to align in more specific directions, the pseudo-grains expressing those directions get a greater weight than the other pseudo-grains. In other words, it means if the inclusions are strongly aligned in the RVE, only few pseudo-grains are required to express such orientation state, those pseudo-grains getting a large weight while the others get a very low one. In summary, all pseudograins together reconstitute the whole RVE (Representative Volume Element) and represent realistically the orientation distribution of the fibers.

The number of pseudo-grains is defined by the number of angle increments (see Orientation). The higher the number of angle increments, the higher the number of pseudo-grains and the thinner they are in terms of angular length, which means it improves the accuracy of the computations. The number of angle increments allowed for FPGF usage ranges from 6 to 16. The suggested number of angle increments for a Digimat to CAE computation with FPGF is 12, it is a good compromise between accuracy and computation time.

Homogenization of Pseudo-grains

An external load applied to the RVE is redistributed over the pseudo-grains, so that each pseudo-grain is in a particular stress/strain state, that depends on its orientation with respect to the load. That means pseudograins (more or less) aligned with the direction of a strain load get a high stress but a low strain in this direction, in comparison with pseudo-grains (more or less) transversely oriented with respect to the direction of a strain load, which then have a smaller stress in this direction but a larger strain. To compute the stress/strain state of one single pseudo-grain, a homogenization step of the matrix response with the inclusion response is first performed. Each pseudo-grain is homogenized separately using the Mori- Tanaka scheme (Level 1). Once this is done, the stress/strain state of the entire RVE (Level 2: macro composite) is computed by homogenizing the pseudo-grains with each other using an iso-strain Voigt scheme. The result is finally the stress/strain prediction of the macro composite, see Figure 9-19, top right.

Application of FPGF Failure Indicators

As discussed above, the stress/strain state of each pseudo-grain is computed separately (Level 1). As a consequence, the FPGF scheme allows the user to apply any kind of failure indicator at the pseudo-grain level.

Failure indicators can be applied to the phases that constitute the pseudo-grains, or simply to the composite a pseudo-grain represents, which is the way the FPGF model should generally be used. The following picture shows how to access the composite and phases FPGF levels. Notice any combination of failure criteria can be defined at the pseudo-grain level.

The application of failure indicators on pseudo-grains means that some pseudo-grains can fail without the necessity that the RVE totally fails. This differs significantly from the usual way of doing which consists of applying failure indicators either on the composite, or on the different phases of which it consists, for the overall RVE.

Advantages of the Pseudo-grain Approach

- Increased resolution for failure detection:

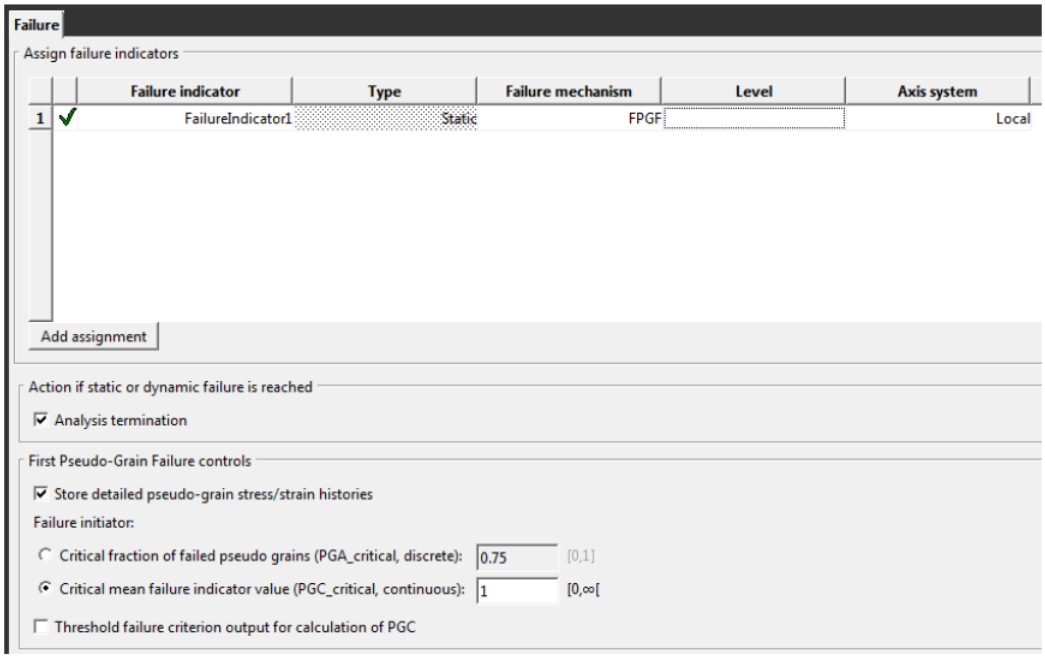


Figure 9-20 When assigning failure indicators on pseudo-grain level (FPGF), the user has the option to choose to store the detailed pseudo-grain stress/strain histories, to choose its failure initiator and the critical value associated to this failure initiator. A threshold options is also available.

The FPGF scheme computes the failure indicator(s) separately for a set of k pseudo-grains (Level 1), rather than computing the failure indicator(s) only once on the macro composite, e.g., the resulting homogenized RVE (Level 2). This results in a higher resolution to detect failure of the RVE, since aligned and transverse pseudo-grains (with respect to the loading direction) are considered separately. Moreover, it allows getting a grey area of the failure status of the RVE, in which failure can be initiated without breaking completely the RVE.

- Simple identification of failure criteria from experimental data:

  Usually, experimental tensile tests on more or less unidirectional dumbbells are performed, which generates some strength thresholds. For most cases, the dumbbells are cut out of an injected plate either along the injection direction (for an aligned dumbbell), or transversely to the injection direction (for a transversely aligned dumbbell). The aligned dumbbell type provides composite tensile strength in the aligned direction xt1 (1 stands for aligned, see Figure 9-21). The second type of dumbbell, in which fibers are transversely oriented with respect to the direction of the load applied, gives an estimate of the composite tensile strength in the transverse direction xt2 (2 stands for transversely aligned, see Figure 9-21).

- A great interest of the FPGF failure indicators comes from how simple it is to define the failure criteria at the pseudo-grain level (Level 1) from the experimental thresholds xt1 and xt2 identified. The xt1 threshold corresponds to the strength at break of the pseudo-grain in the direction of the fibers, and the xt2 threshold corresponds to the strength at break of the pseudo-grain in the direction transverse to the fibers. In other words, an individual pseudo-grain is traduced as a unidirectional composite, which is close to the experimental set-ups which use more or less unidirectional dumbbells.

Remark: Such simplicity can be achieved since the failure criteria are applied using the local axis system. Let's recall that a local axis system is defined by the main direction of the fibers, which actually gives the axis 1 as shown in Figure 9-21. The plane 2-3 is orthogonal to the fibers main direction.

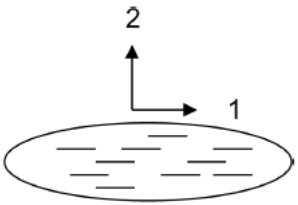


Figure 9-21 Local axis definition in a pseudo-grain.

Figure 9-21 shows local axis definition in a pseudo-grain. Axis 1 is aligned with the (main) fiber direction; axis 2 is perpendicular to axis 1 and lies in the plane of the ply.

FPGF Failure Indicator Outputs (SDV)

To ensure the explanations are clearly understood in the following two sections, about the failure of a single pseudo-grain and failure of a RVE, the FPGF outputs are here already introduced.

No matter how many failure indicators are defined in the material law, there are always one FPGF output, called FPGF. That means if there are more than one failure indicator defined at the pseudo-grain level (FPGF), this output combines the global response of the RVE to all FPGF failure criteria.

The output contains either the value of PGA and PGC that are used to define the failure of the RVE through the use of a critical value called

- Critical fraction of failed pseudo-grains for PGA, and
- Critical mean value of failure criteria for PGC.

The choice between PGA or PGC for the definition of the failure RVE is mutually exclusive. If you choose PGA (resp. PGC), the FPGF output contains the current value of PGA (resp. PGC).

- PGA is the weighted fraction of failed pseudo-grains, over the total weight of pseudo-grains, normalized by its critical value. The RVE is considered unsafe when the output value reaches 1. The output is therefore thresholded to 1.

$$PGA = \frac{\sum_{k=1}^{N} \omega_k F_k}{PGA_{critical}} \quad (9\text{-}109)$$

where:

- N is the total number of pseudo-grains,
- wk is the individual pseudo-grain weight, which reflects its relative contribution to the fiber orientation distribution,
- and F_k is a binary failure flag equal to 0 if the pseudo-grain is sane, and 1 if the pseudo-grain has failed.

The total weight is set to 1.

- PGC is a weighted average value of the failure criterion over the total weight of pseudo-grains (which is set to 1), normalized by its critical value. The main advantage of this output is its continuous evolution.

  This output has slightly different expressions following the "Threshold failure criterion output" option (which is set in the **Failure Assignment** tab).

  - If the option is deactivated (default), PGC is computed as:

    $$PGC = \frac{\sum_{k=1}^{N} \omega_k f_k}{PGC_{critical}} \quad (9\text{-}110)$$

    where f_k is equal to the failure criterion computed for the pseudo-grain k.

  - If the option is activated, PGC is computed as:

    $$PGC = \frac{\sum_{k=1}^{N} \omega_k \max(f_k, 1)}{PGC_{critical}} \quad (9\text{-}111)$$

    which means that the failure criterion of a pseudo-grain is thresholded to 1.

    For the criteria that output several failure indicators (such as the component-based and Hashin criteria), one value is computed for each indicator and the maximum one is retained. For those criteria, the user may find the PGA value more meaningful.

Note that, in the case of a multilayer RVE, the chosen indicator is computed separately for each layer (see Fatigue Models for more information on this point).

Failure of an Integration Point (RVE) - Critical Fraction/Criterion Value Parameters

In Digimat, the failure of a RVE / integration point is modeled such that it can occur although some pseudograins are not yet broken. This is a way to say that at some critical level of failure state in the RVE, the failure initiated can instantaneously propagate in the RVE such that it is from then on considered as totally broken.

The failure status of a RVE can be followed with the main output of the FPGF scheme, FPGF, defined earlier. As soon as one of this output exceed its corresponding critical value, the RVE is reported as broken. In an explicit FE analysis, the FE code triggers the deletion of the integration point if the appropriate check box is checked. In an implicit FE analysis like in Digimat-MF, the analysis is simply stopped.

The failure of the RVE is declared when the critical output reaches 1. The end of the analysis, for Digimat-MF and implicit code except LS-Dyna implicit, or the element deletion, for explicit code and LS-Dyna implicit, can be triggered by FPGF failure. A check box is available for both options in the GUI as shown in Figure 9-22. By default, the end of the analysis is activated in Digimat-MF and deactivated in Digimat-CAE analysis and element deletion is activated in Digimat-CAE analysis. Remark: For multilayer or multiphase RVEs, a specific convention for failure of the RVE is applied, see Multilayer Materials.

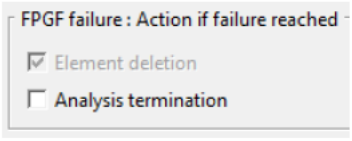


Figure 9-22 Check box is available for the activation/deactivation of the end of the analysis.

Figure 9-22 shows check box is available for the activation/deactivation of the end of the analysis or the element deletion in function of the FEA code.

Microstructure Available for FPGF

As was said earlier, pseudo-grains are strictly 2-phase composites, which might have lead to think that it is unavailable for N-phase composites (N $>$ 2). However Digimat allows to use some specific 3-phase composites, as is described below.

In addition, the FPGF mechanism was in the past limited to monolayer RVEs. As it is explained in the RVE section of the documentation, the multilayer microstructure is very interesting since it brings more accuracy to the computations because of the detailed description of the orientation distribution through the thickness. This multilayer capability is thus also extended to the use of the FPGF mechanism, this is also introduced below.

N-Phases Composites

The First Pseudo-Grain Failure scheme can be applied to some particular multi-phases RVE, i.e., RVE made of a matrix phase reinforced with two inclusion phases. A restriction although applies,

being that the orientation of one of them can be described using an orientation tensor but for the second inclusion, it must be fixed.

Both multi-inclusion homogenization methods, the multi-level method and the multi-step method, can be used to compute the global response of this RVE using FPGF scheme. As for a classical 2-phase composite, failure indicators and FPGF outputs are written in the `*.mac` file or the `*.mtx` file, depending on whether the FPGF failure criteria are respectively applied at the macroscopic level or at the matrix level of pseudo-grains.

Multilayer Materials

The FPGF scheme can be applied to multilayer materials in the same way as it works for monolayer materials. The FPGF failure indicators can be applied either at the composite level or at the phase level of the pseudograins. Since a limitation of FPGF is to define the orientation of the inclusion phase with a tensor, a failure indicator is assigned only to each layer where an orientation tensor describes the orientation of the inclusion's phase.

- RVE failure convention:

  For multilayer material, the failure of the RVE is deemed to occur when the PGA or PGC critical values are reached for all the layers defined in the RVE. This choice was taken to keep consistency with element deletion (especially shell elements) in most FE softwares: the element is removed when all its integration points have failed.

- FPGF outputs:

  If the failure indicator is assigned on the macroscopic level, outputs for all layers where FPGF scheme is applied are written in the macroscopic file (.mac file) with the following convention: FPGFi, where i denotes the number of the layer. A particularity of the multilayer structure is that there are failure statuses, based on the ratio of actual stress/strain of a pseudo-grain over the stress/strain limit, for each layer. These outputs are written in the same file as the other outputs, and with a similar convention, i.e., f1Ai and f1Bi.

  If the failure indicator is assigned to the matrix or to the inclusion phase, failure indicators and FPGF outputs are written in the matrix file (`*.mtx` file) for each layer where the FPGF scheme is applied.

Remarks:

- If FPGF and Standard failure indicators are defined at the same level (phases level or macroscopic level), the failure of the composite is driven by the FPGF model, and not by the standard failure indicator.
- When a standard failure indicator is defined at the macroscopic level, it is computed from the macroscopic fields (i.e., the macroscopic stress or the macroscopic strain depending on the failure indicator definition), and not from the macroscopic field over each pseudo-grain.

Fatigue Models

High cyclic fatigue (HCF) is of great importance to evaluate the lifetime of structures. In the industry, a lot of structures are submitted to a high number of mechanical, thermal or vibration cycles. For the sake of cost reduction, simulation tools are used to predict the lifetime of structures.

To this extent, Digimat offers three phenomenological material models primarily applicable to the lifetime prediction of short fiber reinforced plastics (SFRP) submitted to a high number of cycles. The Pseudo grain fatigue model employs a macroscopic failure indicator at the pseudo grain level, possibly associated to mean stress sensitivity (see subsection Principle). The modified Gerber model is a stress-based fatigue criteria in which mean stress sensitivity is intrinsically taken into account. The energy-based fatigue criterion is a criterion based on the combination of the creep energy density and the cyclic energy density and therefore it also intrinsically embeds the mean stress effects.

The latter two models are applied at the macroscopic level of the composite while the modified Gerber model is combined with a First Pseudo-Grain failure model (FPGF). Both models enable to predict S-N curves for any fiber orientation and multiaxial loadings thanks to limited input in terms of unidirectional S-N curves and constant life diagram (see subsections Usage and Example for more information).

Pseudo Grain Fatigue Model

Principle

The pseudo grain HCF model behaves as if the material macroscopic mechanical state was oscillating between the imposed maximum and minimum stresses σ_{max} and σ_{min} (see Figure 9-23). At the same time, the material strength decreases so that failure is reached after a given number of cycles. This approach does not explicitly model each damage mechanism individually, but captures them on a macroscopic level.

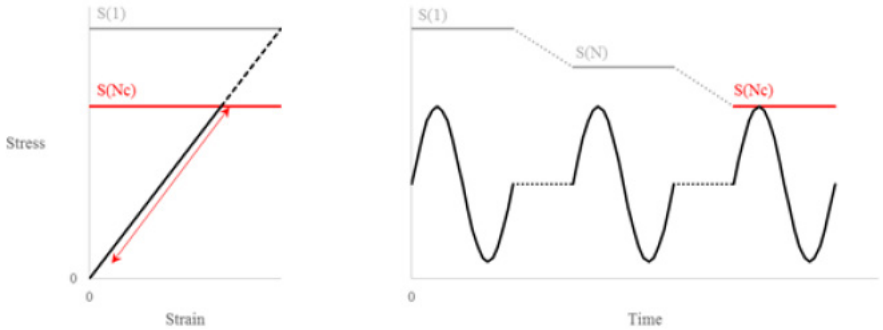


Figure 9-23 Phenomenological pseudo grain HCF model. The model behaves as if the material macroscopic mechanical state was oscillating between the imposed maximum and minimum stresses (left plot). At the same time, the material strength S decreases so that failure is reached after a given number of cycles N_c (right plot).

Such approach makes intuitive the application of a failure indicator whose strength parameters (in terms of stress amplitudes S_a) decrease with the number of cycles. Following the same reasoning, mean stress sensitivity can be accounted for by defining a dependency on the load ratio R for the strength parameters.

$$S_a = \frac{1}{2}(\sigma_{max} - \sigma_{min}) \text{ and } R = \frac{\sigma_{min}}{\sigma_{max}} \tag{9-112}$$

Fatigue Failure Indicator

Multiaxial failure criteria have proved to work accurately in static and dynamic failure analysis of SFRP and can also be exploited to compute the fatigue life of composites (Liu, (2007)). They can be applied in pseudo grains to compute their fatigue life. Pseudo grains consist in virtual 2-phase unidirectional composites in which the actual microstructure is decomposed (see section Distributed Orientations).

For a given macroscopic stress state representative for the cyclic loading amplitude, the failure indicator application proceeds from the following operations.

1. The corresponding strain state is computed by the homogenization procedure. With a viscoelastic matrix (see section Viscoelasticity, this procedure is applied for a given frequency, representative for the cyclic loading.
2. Pseudo grain stresses are computed for this stress state, according to the Voigt model.
3. The Tsai-Hill 3D transversely isotropic criterion is computed for these stresses and a number of cycles estimate N.

$$f(N) = \frac{\sigma_{11}^2}{X^2(N)} - \frac{\sigma_{11}(\sigma_{22}+\sigma_{33})}{X^2(N)} + \frac{\sigma_{22}^2+\sigma_{33}^2}{Y^2(N)} + \left(\frac{1}{X^2(N)} - \frac{2}{Y^2(N)}\right)\sigma_{22}\sigma_{33} + \frac{\sigma_{12}^2+\sigma_{13}^2}{S^2(N)} + \left(\frac{4}{Y^2(N)} - \frac{1}{X^2(N)}\right)\sigma_{23}^2 \tag{9-113}$$

where

- σ_{ij} denotes the components of the stress amplitude in the local axis system associated to each pseudo-grain (e.g. direction 1 corresponds to fiber axis),
- X(N), Y (N) and S(N) denote the axial, in-plane and shear stress amplitudes at failure (a.k.a fatigue strengths) for the number of cycles estimate considered.

4. The average failure indicator at the composite (macroscopic) level is computed for the number of cycles estimate considered. This average is computed according to the pseudo grain weights w_i corresponding to the pseudo grain decomposition considered equivalent to the microstructure (orientation tensor) considered.

$$f_{composite}(N) = \sum_{i=1}^{n} w_i f_i(N) \tag{9-114}$$

5. The critical number of cycles N_c is computed. It is determined iteratively by varying the number of cycles estimate N until

$$f_{composite}(N_c) = 1 \tag{9-115}$$

The above-described failure indicator application reduces in operations 3 and 5 for unidirectional composites. Indeed such composites consist in single pseudo grain microstructures.

Mean Stress Sensitivity

Mean stress sensitivity refers to the variability of the apparent material strength to the mean stress S_m around which the cyclic loading is performed.

$$S_m = \frac{1}{2}(\sigma_{max} + \sigma_{min}) \tag{9-116}$$

Indeed this strength varies, e.g., due to creep or compressive effects, so that different stress amplitudes yield the same number of cycles for different mean stresses. Such sensitivity is usually depicted in the shape of Constant Life Diagrams (CLD; see Figure 9-24). These diagrams actually contain the same information than several S-N curves usually measured for constant load ratios (more than constant mean stresses). These S-N curves appear on slant lines extending from the origin (https://www.youtube.com/watch?v=C5OaaQlcVxU).

Indeed, stress amplitudes and mean stresses relate through the following formula.

$$S_a = \frac{1-R}{1+R} S_m \tag{9-117}$$

Composite CLD do not necessarily exhibit the features classically expected for metals. In particular, constant life lines do not converge towards an equal mean stress when decreasing the stress amplitude (R -->1). Indeed, according to the simplest intuitive reasoning, the largest affordable mean stress for a cyclic test until failure is the UTS, for which the stress amplitude vanishes. However such reasoning is overly optimistic for SFRP: a long-term test with a vanishing stress amplitude is more comparable with a creep test – associated to strengths smaller than the UTS – than with a monotonic tensile test until failure. Consequently the extrapolations of constant life lines towards $R = 1$ are more comparable with creep strengths for which the time to failure is substituted with an equivalent number of cycles (Mallick and Zhou, (2004)). Then the most simple CLD exhibits parallel constant life lines.

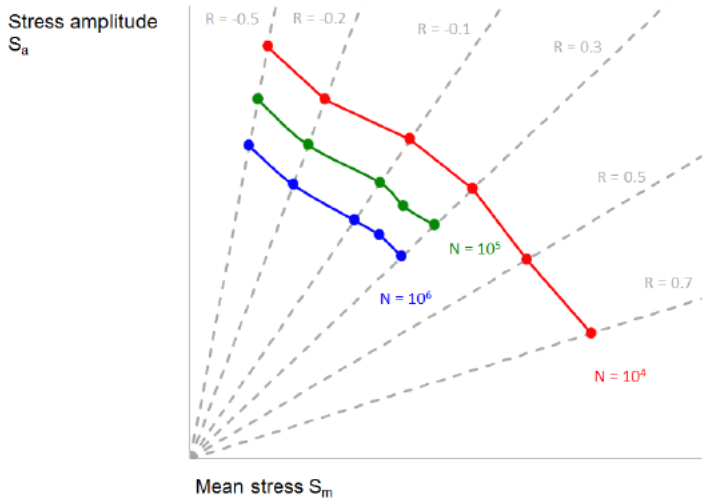


Figure 9-24 Sample constant life diagram for a glass fiber reinforced polyamide.

Figure 9-24 shows sample constant life diagram for a glass fiber reinforced polyamide (Robert et al., (2015)). Constant life lines do not converge towards an equal mean stress when decreasing the stress amplitude.

To account for mean stress sensitivity in the above described failure indicator, its strength parameters receive an additional dependency on the load ratio R, an independent variable contrary to the mean stress (Hashin,(1981)). Moreover, the above reported simplest CLD suggests to uncouple N and R dependencies and to parametrize the failure indicator via a stress amplitude multiplier $\mu(R)$ in addition to the unidirectional S-N curves at a reference load ratio R_{ref}.

$$S(N, R) = \mu(R)S(N, R_{ref}) \tag{9-118}$$

Finally a pragmatic formulation of the stress amplitude multiplier consists in a piecewise function yielding a piecewise linear CLD (Vassilopoulos et al., (2010)).

When the load ratio is not explicitly defined, it is computed from the mean stress and the stress amplitude, e.g., for a uniaxial loading.

$$R = \frac{S_m - S_a}{S_m + S_a} \tag{9-119}$$

With a viscoelastic matrix, the mean stress is computed using the asymptotic stiffness. Hence the long term behavior represented by the viscoelastic model influences mean stress sensitivity.

Usage

A pseudo grain fatigue analysis in Digimat (see Figure 9-25) requires the definition of:

- a fatigue failure indicator, itself consisting of:
  - base strength parameters
  - optionally, N_c dependencies over these strength parameters,
  - optionally, a mean stress sensitivity function (e.g., a constant life diagram);
- a fatigue failure assignment
- a multilayer failure trigger definition (optional)
- underlying analysis items.

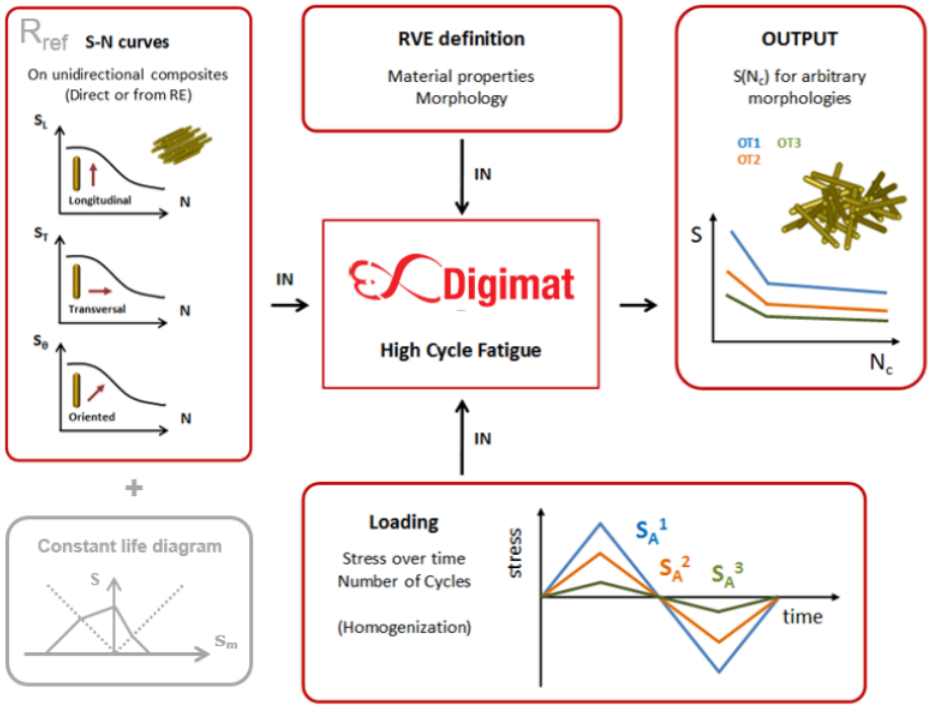


Figure 9-25 Pseudo grain fatigue analysis definition in Digimat.

Fatigue Failure Indicator Definition

The Digimat fatigue failure indicator definition derives from the definition of the stress-based Tsai-Hill 3D transversely isotropic failure indicator (see Tsai-Hill 3D Transversely Isotropic), where strengths are replaced by Nc-dependent critical stress amplitudes. So it typically requires the definition of 3 strength parameters (see Figure 9-26):

- The axial tensile strength X, a.k.a the longitudinal (0° loading) critical stress amplitude for a unidirectional composite.

- The in-plane tensile strength Y, a.k.a the transverse (90° loading) critical stress amplitude for a unidirectional composite.
- The transverse shear strength S, a.k.a the shear-12 critical stress amplitude for a unidirectional composite.

These S-N curves are (optionally) defined through the regular strength dependency assignment workflow, where the dependency parameter is the critical number of cycles (N_c) and the dependency relationship is described by a piecewise log-linear function (see Figure 9-26). These functions act as multipliers over their reference strength parameter.

Each of these function must be strictly decreasing in their initial range of definition, i.e., they must satisfy the following condition:

$$\frac{(S_a^{i+1} - S_a^i)}{S_a^{i+1} \times (\log N_c^{i+1} - \log N_c^i)} \leq -\alpha_{min} \tag{9-120}$$

The default value for min is 10^{-3}, which means that the stress amplitude decreases by 0.1% per decade. It is possible to change this default value, or even to completely deactivate these checks, through the **Minimum relative slope of the S-N curves** checkbox in the Integration parameters tab (see subsection High Cycle Fatigue Controls).

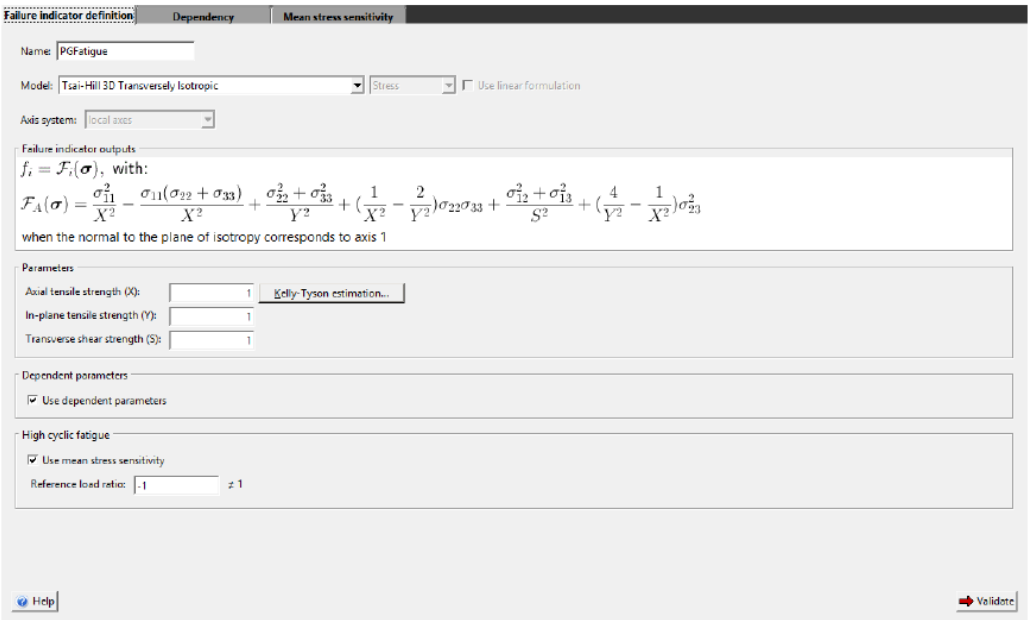


Figure 9-26 Fatigue failure indicator definition (base strength parameters).

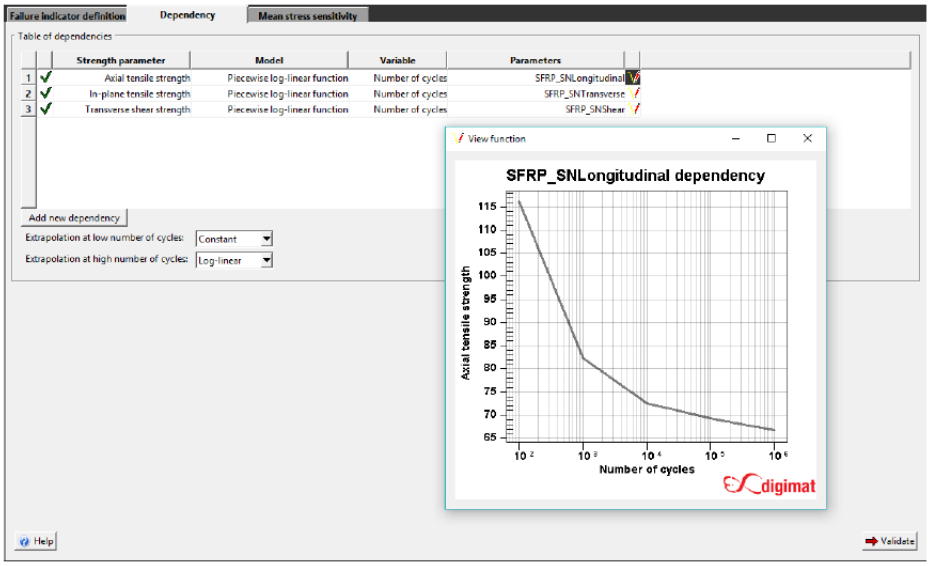


Figure 9-27 Fatigue failure indicator definition (N_c dependencies).

In addition to the actual function assignment, the S-N curves are associated with extrapolations applied outside the widest number of cycles range across all S-N curves, within which log-linear interpolation or extrapolation are applied.

Finally, a mean stress sensitivity definition may optionally be assigned to the fatigue failure indicator, by:

- checking the **Use mean stress sensitivity** button in the Failure indicator definition tab;
- defining the **reference load ratio** in the Failure indicator definition tab;
- defining the additional parameters in the Mean stress sensitivity tab (as described hereafter).

Mean Stress Sensitivity Definition

The mean stress sensitivity definition mainly consists in a constant life diagram (CLD) representative for the behavior of a unidirectional composite (see Figure 9-28).

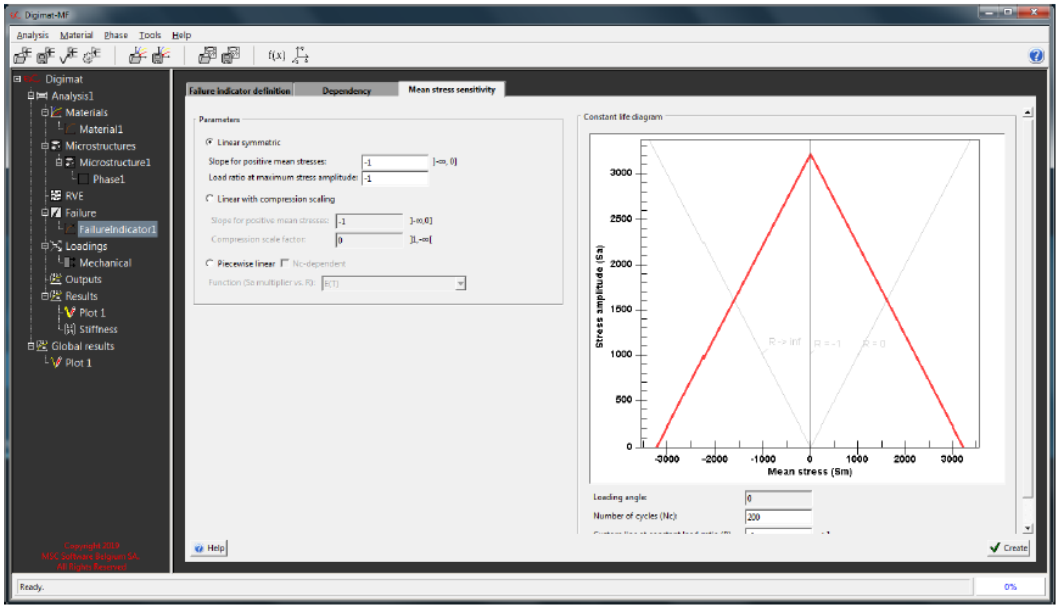


Figure 9-28 Mean stress sensitivity definition mainly consists of a constant life diagram.

Digimat offers several simplified CLD types, in addition to the general piecewise linear one, corresponding to different sets of input parameters.

- A linear symmetric CLD exhibits 2 regimes:
  - a linear decrease of the stress amplitude with the mean stress with a constant (negative) slope when R increases from a given value (at maximum stress amplitude) towards 1;
  - and a symmetric evolution with the mean stress outside this R interval.
- A linear CLD with compression scaling exhibits 3 regimes:
  - a linear decrease of the stress amplitude with the mean stress at a constant (negative) slope when R increases from -1 towards 1;
  - an offseted decrease from the same maximum stress amplitude towards 0 when the mean stress decreases towards a minimum corresponding to the maximum mean stress multiplied by a given compression scale factor;
  - and a constant (maximum) stress amplitude in the remaining interval.

- A piecewise linear CLD is parametrized after a function relating the stress amplitude multiplier to R, imported via the function manager. The CLD can be only load ratio dependent through 1D function, or also dependent of the number of cycles thanks to 2D function (see Functions). The later case is available only if the N_c-dependent option is toggled on. The 2D function must be defined such that X1 stands for the load ratio R, and X2 stands for the number of cycles Nc (with a piecewise log-linear interpolation over N_c). See Figure 9-29 for an example of suitable 2D function definition.

  According to Equation , such a function multiplies the unidirectional S-N curves to account for mean stress sensitivity at the pseudo grain level. Hence, it needs associating a stress amplitude multiplier equal to 1 to the load ratio of the unidirectional S-N curves. It exhibits linear evolutions of the stress amplitude with the mean stress between each successive values of R defined in the function. In addition, it extrapolates linearly the stress amplitude towards 0 outside this range with a slope deduced from the 2 adjacent points below (resp. above) $R = 1$.

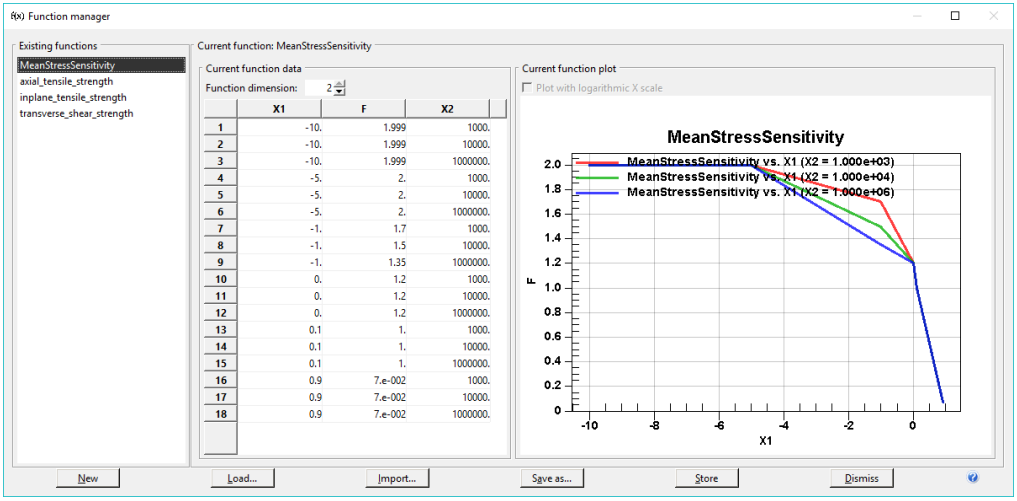


Figure 9-29 Example of function definition suitable for 2D mean stress sensitivity definition.

Fatigue Failure Indicator Assignment

Just like regular failure indicators, fatigue failure indicators shall be assigned through the Failure assignment tab (see subsection Failure Indicator Assignment; although this assignment is pretty trivial since the failure type, mechanism, level and axes are enforced.

Additionally, if the RVE is a multilayer, a multilayer failure trigger can be defined, just like for regular failure criteria (see subsection Failure for Multilayer RVE).

Underlying Analysis Definition

Besides the fatigue failure indicator, optionally mean stress sensitive, a pseudo grain fatigue analysis requires the definition of classical analysis items:

- (visco)elastic materials,
- a 2-phase microstructure,
- a single-layer or multilayer RVE,
- and a cyclic stress loading (see section Fatigue Loading).

The Graphical interface enables to define a Pseudo-grain fatigue criteria on top of an elastoplastic model. When this combination is used inside Digimat product, the material model will automatically be downgraded in to an elastic model. When this combination is used inside CAEfatigue or nCode, the elastoplastic behavior will be used to compute the plasticity correction.

The cyclic stress loading consists in the application of the failure indicator for a set of numbers of cycles or stress amplitudes defined as input. It does not really involve several computations for discretized time histories of stresses at constant amplitude between minimum and maximum values until a critical number of cycles. Yet it yields S-N curves considered equivalent to such computations, in terms of the stress amplitude S_a at a predefined load ratio R. For a mean stress insensitive model (**Use mean stress sensitivity** unchecked), this load ratio corresponds to the one of the unidirectional S-N curves defining the fatigue failure indicator.

Remark: When the S-N curve is calculated for a range of stress amplitudes, a dichotomy algorithm is used to retrieve the corresponding critical number of cycles. The parameters ruling this algorithm can be edited in the **High cycle fatigue controls** section of the Integration parameters tab (see subsection High Cycle Fatigue Controls). The **dichotomy algorithm tolerance** defines the (unitless) tolerance target in terms of average failure indicator. The **dichotomy algorithm number of iterations** defines the maximum number of iterations of the algorithm before non-convergence is declared.

Example

Figure 9-30 shows an example of computed S-N curves for different microstructures based on the described approach. The input S-N curves are also shown (red, green and blue).

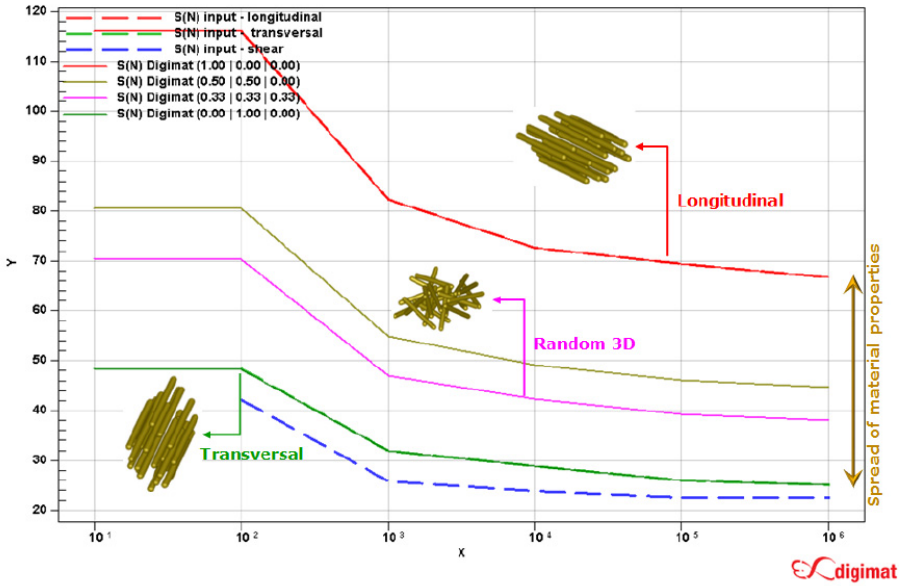


Figure 9-30 Computed S-N curves for different microstructures based on the definition of number of cycles.

Modified Gerber Model

The second phenomenological fatigue model in Digimat is the modified Gerber model.

A modified form of the known stress-based Gerber criterion was proposed by Mallick and Zhou, (2004) allowing to account for the effect of mean stress on fatigue strength of short E-glass fiber reinforced polyamide-66 composites.

The Gerber equation was reformulated in terms of the ultimate tensile strength σ_{UTS} using the fatigue strength factor α and creep rupture strength factor β to obtain the following non-dimensional equation:

$$\frac{S_a}{\alpha\sigma_{UTS}} + \left[\frac{S_m}{\beta\sigma_{UTS}}\right]^2 = 1 \qquad (9\text{-}121)$$

Where S_a denotes the stress amplitude, and S_m is the mean stress.

As shown in Prashanth et al., (2020), the normalization by σ_{UTS} allows to unify with a single set of parameters the fatigue data with respect to fiber orientation and atmospheric condition.

Indeed, this criterion was applied by Prashanth et al., (2020) on a wide fatigue database obtained on Polyamide 66 reinforced with 50% of glass for an extended range of load ratios (from -0.5 to 0.7), three orientations from the injection direction (0°, 45°, 90°) and two environmental conditions (50% humidity ratio and 80°C, 80% humidity ratio and 23°C).

The model showed a rather good unification of the fatigue data (over 80% of database is unified within a scatter of factor 5) with a unique set of parameters.

Parameters α and β are optimized to better fit to the full database and take the following power-law form:

$$\alpha = \alpha_1 N_c^{\alpha_2} \tag{9-122}$$

$$\beta = \beta_1 N_c^{\beta_2} \tag{9-123}$$

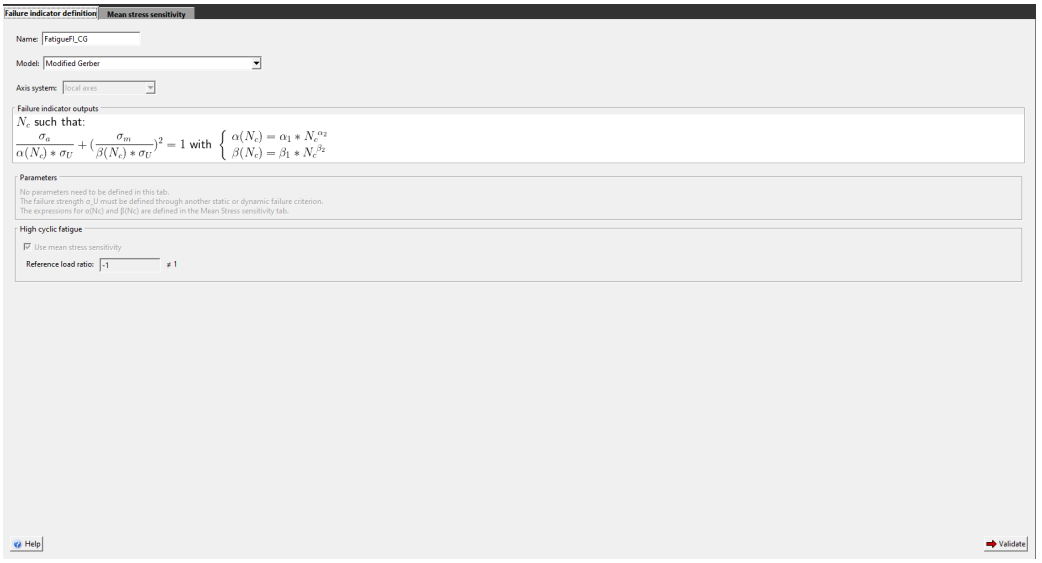
Usage

The modified Gerber model in Digimat behaves as if the material macroscopic mechanical state was oscillating between the imposed maximum and minimum stresses σ_{max} and σ_{min}.

Unlike the Pseudo grain fatigue model that is applied at the pseudo-grain level, the modified Gerber model is applied at the composite level but should be used in conjunction with a static first Pseudo-Grain failure model (FPGF) in order to get an accurate estimation of the ultimate tensile strength.

It requires the definition of:

- Classical analysis items:
  - materials: the model is intended for (visco) elastic (visco) plastic matrix reinforced with elastic fibers
  - a 2-phase microstructure
  - a single-layer RVE: multilayer RVEs are not supported for this model
  - a cyclic stress loading (see section Fatigue Loading).
- A fatigue failure indicator with mean stress sensitivity parameters identified from fatigue database (see Figure 9-31).
- A static failure indicator to be defined at the pseudo-grain level using the FPGF scheme with the option **Use linear formulation** activated (see Figure 9-32).



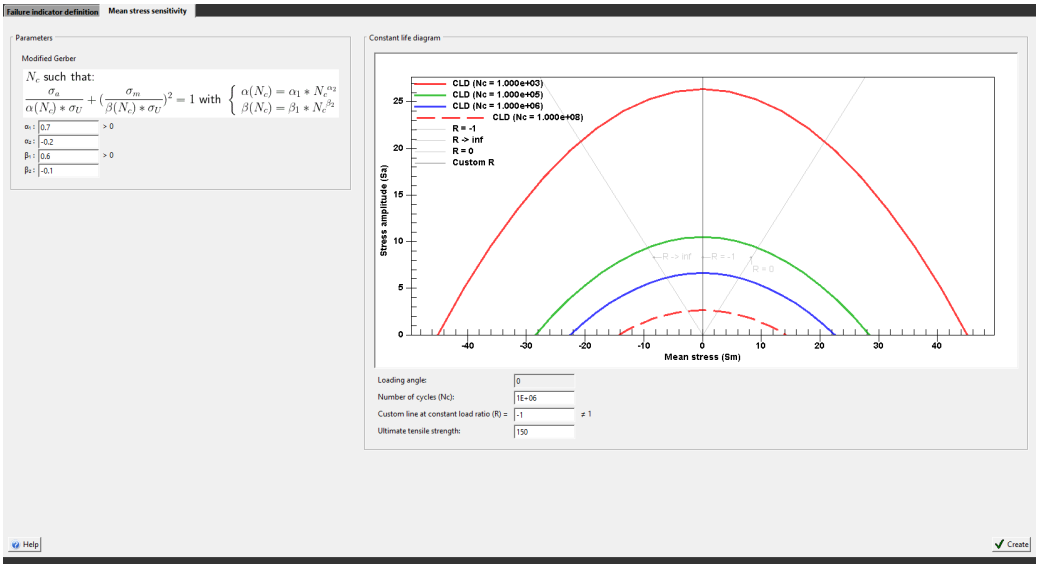


Figure 9-31 The modified Gerber model definition and the associated CLD visualization.

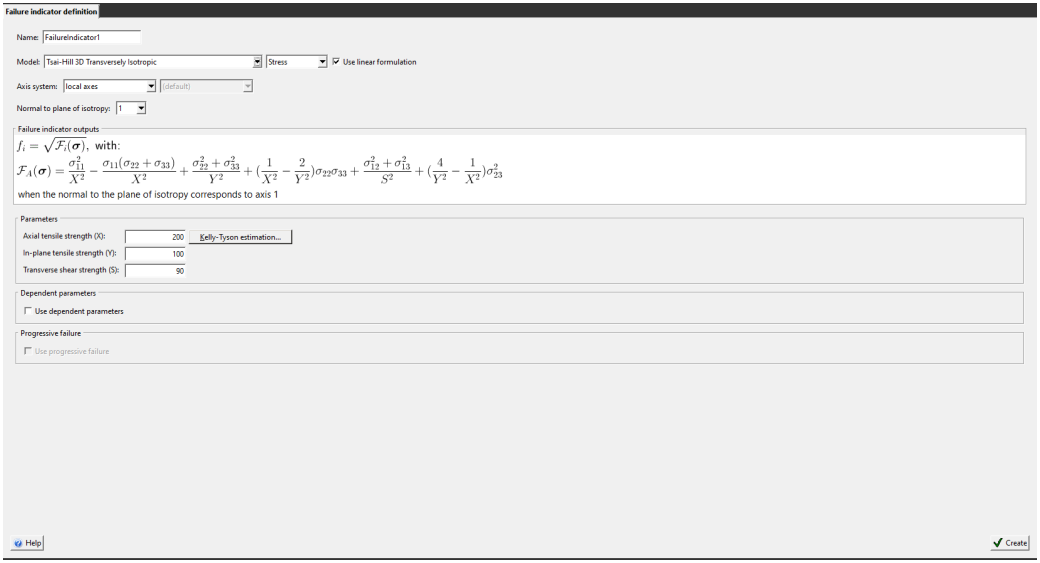


Figure 9-32 Failure indicator definition window: the "Use linear formulation" box should be checked.

For a given macroscopic stress state representative for the cyclic loading amplitude, the life time computation procedure consists of the following 2 steps:

1. The ultimate tensile strength is estimated based on the defined microstructure and the assigned static failure model.
2. The number of cycles to failure is estimated for the given stress amplitude and predefined load ratio R from the modified Gerber equation.

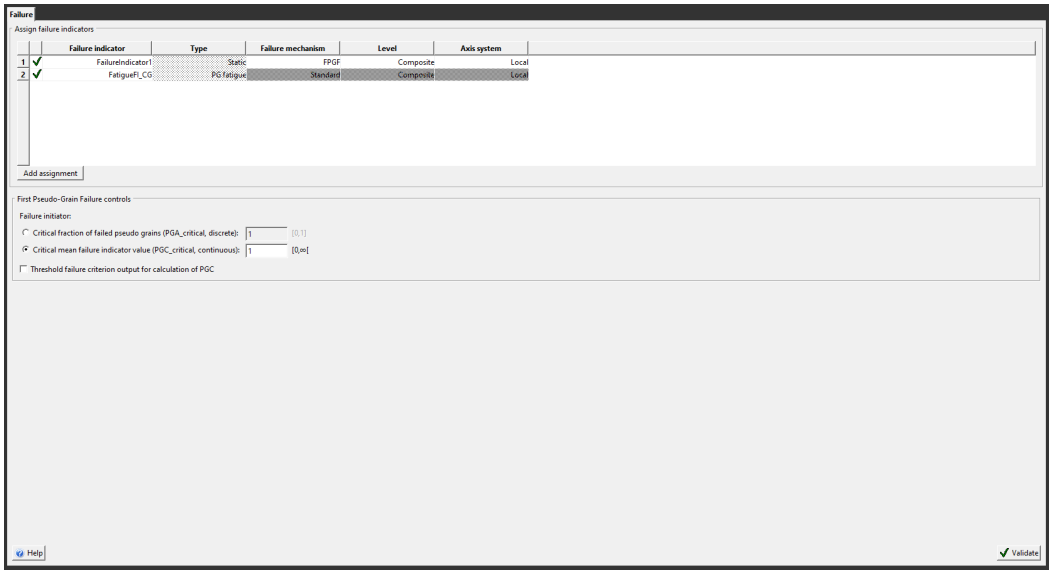


Figure 9-33 Static failure indicator should be assigned as a FPGF failure mechanism along with the fatigue criterion.

Energy-based Criterion

The energy-based fatigue criterion is a criterion based on a mixed formulation that combines two energetic quantities that are the creep energy density and the cyclic energy density.

It has been shown in recent studies (Raphael et al., (2019), Prashanth et al., (2020)) that the creep energy (associated with the ratcheting effect) and the cyclic energy are among the most relevant indicators for the estimation of the fatigue life for respectively positive and negative load ratios. Combining both cyclic features allow to accurately describe the fatigue data for both positive and negative load ratios, different loading orientations with respect to the injection direction, and environment conditions with a single set of parameters.

For a given i^{th} cycle, the cyclic energy density is computed as follows:

$$W_c = \frac{1}{2}\Delta\sigma \cdot \Delta\varepsilon \tag{9-124}$$

where, $\Delta\sigma$ and $\Delta\varepsilon$ are the amplitudes of the variations of nominal stress and strain over the cycle, respectively.

The creep energy density can be written as:

$$W_{cr} = \sigma_m \frac{d\varepsilon_m}{dN_c} \tag{9-125}$$

Where σ_m and ε_m are the mean nominal stress and strain over the cycle, respectively.

The original version of the energy based model was proposed by Prashanth et al., (2020).

It determines the number of cycles to failure as:

$$\frac{1}{N_c} = \left(\frac{W_{cr}^*}{\alpha_1}\right)\alpha_2 + \left(\frac{W_c^*}{\beta_1}\right)\beta_2 \tag{9-126}$$

Where, the energetic quantities W_{cr}^* and W_c^* are measured at mid-life.

The four parameters α_1, α_2, β_1 and β_2 are optimized to minimize the discrepancy between the estimated and experimental fatigue lifetime over the entire experimental database.

This criterion was applied by Prashanth et al., (2020) on a wide fatigue database obtained on Polyamide 66 reinforced with 50% of glass for an extended range of load ratios (from -0.5 to 0.7), three orientations from the injection direction (0°, 45°, 90°) and two environmental conditions (50% humidity ratio and 80°C, 80% humidity ratio and 23°C).

It showed particularly good unification of the fatigue data: 96.5% of database is unified within a scatter of factor 3, with a unique set of parameters.

A modified version of the criterion was developed later by the same authors in order to make the model more suitable for software usage in industrial applications.

As alternative to energies at mid-life, stabilized cyclic energy values $W_{c_{sta}}$ and initial value of creep energy W_{cr_0} and its slope, with respect to N_c, $W_{cr_{sl}}$ can be used to solve the criterion, leading to the following expression:

$$\frac{1}{N_c} = \left(\frac{W_{cr_0} N_c^{W_{cr_{sl}}}}{\alpha_1}\right)^{\alpha_2} + \left(\frac{W_{c_{sta}}}{\beta_1}\right)^{\beta_2} \tag{9-127}$$

Usage

The energy-based fatigue model requires the definition of:

- Classical analysis items:
  - materials: strain rate elastoplastic or viscoelastic-viscoplastic matrix reinforced with elastic fibers;
  - a 2-phase microstructure;
  - a single-layer RVE: multilayer RVEs are not supported for this model;
  - a cyclic stress loading (see section Fatigue Loading).
- A fatigue failure indicator with parameters identified from fatigue database (see Figure 9-34).

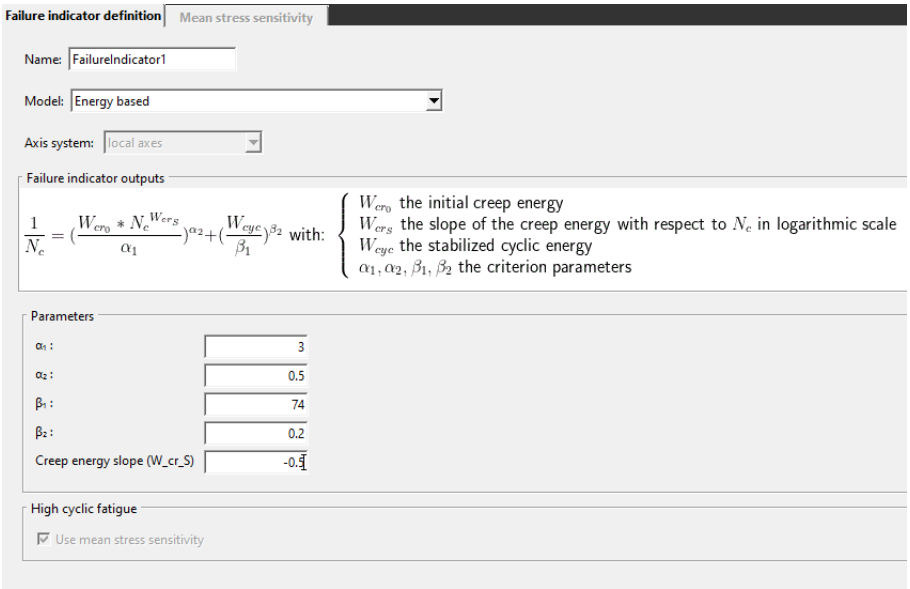


Figure 9-34 The energy-based fatigue model definition window.

Matrix Damage Fatigue Model

High cyclic fatigue is of great importance to evaluate the lifetime of structures. In the industry, a lot of structures are submitted to a high number of mechanical, thermal or vibration cycles. For the sake of cost reduction, simulation tools are used to predict the lifetime of structures. The damage model at the matrix scale for high cyclic fatigue implemented in Digimat is based on the two scale damage model proposed by Desmorat et al. (2007) for thermo-mechanical composites. This model is based on the idea that damage occurs at the microscopic scale, i.e., a scale smaller than the mesoscopic one of the RVE. The key idea of this model is that for high cyclic fatigue (or for quasi-brittle materials), damage and plasticity occurs at the micro-scale and have no influence on the macroscopic thermo-elastic behavior except for failure. This leads to consider a micro-mechanics based model of a weak microscopic inclusion subjected to plasticity and damage embedded in a thermo-elastic mesoscopic RVE. The scale transition between the mesoscopic scale and the microscopic scale is governing by the modified Eshelby-Kröner localization law.

The objective of this documentation section is to describe how Digimat treats the lifetime prediction of composite material submitted to a high number of cycles.

The matrix damage fatigue model is proposed as a material model that can be assigned to the matrix. It is currenlty supported for two types of microstructure

- unidirectional (i.e two phases composites with continuous fiber inclusions having a fixed orientation)

- fabric with basic yarns inclusions.

Some guidelines and limitations will be proposed to help the user to define his/her own fatigue model.

Theory

In this section, the two scale model proposed by Desmorat et al. (2007) will be presented. We will restrict our attention to isothermal behavior at each scale. This mean that thermal aspect won't be taken into account.

General Concept of the Two-scale Model

At the mesoscopic scale, the elastic law for the RVE reads:

$$\varepsilon^e = \frac{1+\nu}{E}\sigma - \frac{\nu}{E}\mathrm{Tr}\sigma 1 \tag{9-128}$$

At the microscopic scale, an elasto-plastic law coupled with damage is considered. No viscosity is considered in this model. The set of constitutive equation at the microscopic scale is defined as follow:

$$\begin{aligned}
\varepsilon_\mu &= \varepsilon_\mu^e + \varepsilon_\mu^p \\
\varepsilon_\mu^e &= \frac{1+\nu}{E}\bar{\sigma}_\mu - \frac{\nu}{E}\mathrm{Tr}\bar{\sigma}_\mu 1 = (a^e)^{-1} : \bar{\sigma}_\mu \\
\dot{\varepsilon}_\mu^p &= \frac{3}{2}\frac{\bar{S}_\mu - X_\mu}{(\bar{\sigma}_\mu - X_\mu)_{eq}}\dot{p}_\mu \\
\frac{d}{dt}\left(\frac{X_\mu}{C_y}\right) &= \frac{2}{3}\dot{\varepsilon}_\mu^p(1-D) \\
\dot{D} &= \left(\frac{Y_\mu}{S}\right)^q \dot{p} \quad \text{if } p_\mu > p_\mu^D \ \text{(the damage threshold)}
\end{aligned} \tag{9-129}$$

with C_y the plastic modulus, S the damage strength and q the damage exponent. $\bar{s}_\mu$ stands for the deviatoric part of $\bar{\sigma}_u$ and a^e, for the elastic stiffness matrix. In the yield criterion, X_μ stands for the kinematic linear stress at the microscopic scale and the yield stress depends on the asymptotic fatigue limit of the material, denoted σ_f^∞

$$f_\mu = (\bar{\sigma}_\mu - X_\mu)_{eq} - \sigma_f^\infty \tag{9-130}$$

with

$$\bar{\sigma}_\mu = \frac{\sigma_\mu}{1-D} \tag{9-131}$$

The elasticity domain is defined by $f_\mu < 0$. A crack is initiated when D reaches the critical damage value D_c. The function Y_μ (also called elastic energy density) which is defined in the damage evolution equation , reads as follow:

$$Y_\mu = \frac{1+\nu}{2E}\left[\frac{\langle\sigma_\mu\rangle^+ : \langle\sigma_\mu\rangle^+}{(1-D)^2} + h\frac{\langle\sigma_\mu\rangle^- : \langle\sigma_\mu\rangle^-}{(1-hD)^2}\right] - \frac{\nu}{2E}\left[\frac{\langle Tr\sigma_\mu\rangle^2}{(1-D)^2} + h\frac{\langle -Tr\sigma_\mu\rangle^2}{(1-hD)^2}\right] \tag{9-132}$$

where $\langle\sigma\rangle^+$ and $\langle\sigma\rangle^-$ denote respectively the positive and the negative part of the stress tensor in terms of principal values and $\langle x\rangle$ stands for the positive part of the scalar x, i.e., $\langle x\rangle = \max(x, 0)$. This definition leads to a damage evolution which is smaller in compression than in tension due to the micro-defects closure parameters h.

The set of equations which governs the damage at the microscopic scale is fully nonlinear and the damage evolution is given by a differential equation.

The transition from mesoscopic scale to the microscopic one is governed by the Eshelby's-Kröner localization law (or the localization law of the self-consistent scheme - see for example Berveiller and Zaoui (1979) or Kröner (1961)):

$$\sigma_\mu = \sigma - 2\mu(1-b)\varepsilon_\mu^p \tag{9-133}$$

where b is given by a decomposition of the Eshelby's tensor into a spherical and a deviatoric part such that respectively $S_E : 1 = a1$ and $S_E : x^D = bx^D$ where x^D denotes any deviatoric tensors. For spherical inclusions, parameters a and b have the following definition:

$$\begin{cases} a = \dfrac{1+\nu}{3(1-\nu)} \\ b = \dfrac{2}{15}\dfrac{(4-5\nu)}{(1-\nu)} \end{cases} \tag{9-134}$$

This principle leads in the case of damaged model to the following definition of the strain at the microscopic scale from the strain at the mesoscopic scale replacing σ and σ_μ in equation by their expression given respectively in equation and equation :

$$\varepsilon_\mu = \frac{1}{1-bD}\left\{\varepsilon + \frac{(a-b)D}{3(1-aD)}Tr\varepsilon 1 + b[(1-D)\varepsilon_\mu^p - \varepsilon^p]\right\} \tag{9-135}$$

As in Digimat, we assume an elastic behavior at the mesoscopic scale, it then follows the definition of the microscopic strain field:

$$\varepsilon_\mu = \frac{1}{1-bD}\left\{\varepsilon + \frac{(a-b)D}{3(1-aD)}Tr\varepsilon 1 + (1-D)b\varepsilon_\mu^p\right\} \tag{9-136}$$

For an homogeneous material, the mesoscopic strain ε is assimilated to the macroscopic strain $\bar{\varepsilon}$ imposed on the boundary of the RVE. But for a composite material, the mesoscopic ε strain is assimilated to the strain in the matrix phase which is related to the macroscopic strain through the following relation regarding the Mori-Tanaka homogenization procedure:

$$\varepsilon = \langle \varepsilon \rangle_{w0} = (B^{\varepsilon})^{-1} : A^{\varepsilon} : \bar{\varepsilon} \tag{9-137}$$

Over a time increment $[t_n, t_{n+1}]$, once the strain field is known at the mesoscopic strain at time t_{n+1}, the microscopic fields at time t_{n+1} are solution of the following nonlinear set of equation:

$$\begin{cases} \varepsilon_{\mu}(t_{n+1}) = \varepsilon_{\mu}^{e}(t_{n+1}) + \varepsilon_{\mu}^{p}(t_{n+1}) \\ \qquad = \dfrac{1}{1 - bD(t_n)} \left\{ \varepsilon(t_{n+1}) + \dfrac{(a-b)D(t_n)}{3[1 - aD(t_n)]} Tr\varepsilon(t_{n+1}) 1 + [1 - D(t_n)] b\varepsilon_{\mu}^{p}(t_{n+1}) \right\} \\ \varepsilon_{\mu}^{e}(t_{n+1}) = (a^{e})^{-1} : \bar{\sigma}_{\mu}(t_{n+1}) \\ \Delta\varepsilon_{\mu}^{p} = \dfrac{3}{2} \dfrac{\tilde{S}(t_{n+1}) - X_{\mu}(t_{n+1})}{(\bar{\sigma}_{\mu}(t_{n+1}) - X_{\mu}(t_{n+1}))_{eq}} \Delta p_{\mu} \\ \Delta X_{\mu} = \dfrac{2}{3} C_y [1 - D(t_n)] (\Delta\varepsilon_{\mu}^{p}) \\ \Delta D = \left(\dfrac{Y_{\mu}}{S} \right)^{q} \Delta p \quad \text{if } p_{\mu} > p_{\mu}^{D} \text{ (the damage threshold)} \end{cases} \tag{9-138}$$

Desmorat et al. (2007) proposes an implicit scheme to perform the time integration of the two scale set of nonlinear equations . The main advantage of this algorithm is that the fields at the microscopic scale are computed without any iteration. The cycle is applied on the boundary of the RVE once the damage parameter D reaches the critical damage parameter D_c. In this case, the matrix is assumed to be broken and the analysis stops.

Jump in Cycle Procedure

For periodic loadings, when a large number of cycles is needed to reach the failure of the material, a step by step procedure in time becomes prohibitive in term of CPU time especially when the critical number of cycles becomes large (104, 108, etc.). A simple jump in cycle procedure has been proposed by Lemaitre and Doghri (1994) in order to find the critical number of cycles jumping a block of cycles. This procedure is described in the following section.

A first order Taylor expansion of the accumulated plastic strain and of damage evolution parameter around any number of cycles $(N + \Delta N)$ gives:

$$\begin{cases} p(N + \Delta N) \approx p(N) + \dfrac{\partial p}{\partial N}(N)\Delta N \\ D(N + \Delta N) \approx D(N) + \dfrac{\partial D}{\partial N}(N)\Delta N \end{cases} \tag{9-139}$$

where $\frac{\partial p}{\partial N}$ and $\frac{\partial D}{\partial N}$ denote respectively the increment of the accumulated plastic strain and the increment of the damage parameter over the cycle N. This approximation assumes that during the jump of N, the accumulated plastic strain and the damage parameter evolve linearly with respect to N. The block of cycles to jump, N, is unknown and is defined as follow from equation:

$$\begin{cases} \Delta N \approx \dfrac{p(N+\Delta N) - p(N)}{\left(\dfrac{\partial p}{\partial N}\right)} = \dfrac{\Delta p}{\left(\dfrac{\partial p}{\partial N}\right)} \\ \Delta N \approx \dfrac{D(N+\Delta N) - D(N)}{\left(\dfrac{\partial D}{\partial N}\right)} = \dfrac{\Delta D}{\left(\dfrac{\partial D}{\partial N}\right)} \end{cases} \tag{9-140}$$

where quantities p and D have to be defined by the user. These values determine the accuracy of the procedure. Lemaitre and Doghri (1994) propose the following approximation to evaluate p and D:

$$\Delta D = \frac{D_c}{n} \rightarrow \Delta p = \left(\frac{S}{Y(\Delta\bar{\sigma})}\right)^q \Delta D \tag{9-141}$$

where n is a use-defined parameter ($n = 50$ by default). ΔD is first estimated and Δp is then deduced from D using the damage evolution law. The function Y is computed from $\Delta\bar{\sigma}$ which stands for the effective stress amplitude at the cycle N: $\Delta\bar{\sigma} = \frac{(\bar{\sigma}_{max} - \bar{\sigma}_{min})}{2}$. The block of N cycles to jump is finally defined as follow:

$$\Delta N = \min\left\{\frac{\Delta p}{\left(\dfrac{\partial p}{\partial N}\right)}, \frac{\Delta D}{\left(\dfrac{\partial D}{\partial N}\right)}\right\} \tag{9-142}$$

The jump of cycle can finally be summarized as follow:

$$\text{if } p > p_D \text{ then } \begin{cases} \text{Do a cycle to compute: } \dfrac{\partial p}{\partial N}, \dfrac{\partial D}{\partial N} \text{ and } \Delta\bar{\sigma} \\ \text{Compute } \Delta p \text{ and } \Delta D \text{ from equation 9-132} \\ \text{Compute the number of block of cycle } \Delta N \text{ from equation 9-133} \\ \text{Update variables } p(N+\Delta N) \text{ and } D(N+\Delta N) \text{ from equation 9-130} \end{cases} \tag{9-143}$$

As this approach is based on an heuristic concept, it may happen in some case that the jump in cycle procedure doesn't work correctly. Then, the user can change the value of n in the **High cycle fatigue controls** section of the Integration parameters tab (see Subsection High Cycle Fatigue

Controls). But experience shows that the default n = 50 leads to very good results, with acceptable CPU times.

Model Parameters

The fatigue behavior is defined by the parameters of the High Cycle Damage material used at the matrix level:

- E, ν and Cy (Young's modulus, Poisson's ratio and kinematic hardening modulus): these parameters are identified on a monotonic tensile curve.
- The critical damage Dc: usually this parameter is lower than 1 and, for example, is equal to 0.3 for metals.
- The damage initiation threshold pD: this positive or null parameter stands for the accumulated plastic level at which damage can occur.
- The micro-defects closure parameter h $(0 \le h \le 1)$: the value of this parameter is directly related to the nature of the material. For example, for metal $h \approx 0.2$.
- S, q and σ_f^∞ (damage strength, damage exponent and fatigue limit): these parameters can be identified on the Whöler curve of the material.

Units

Digimat-MF is not bound to a particular system of units. Nevertheless, units must be consistent throughout the definition of the material model, e.g., if the Young's modulus is given in MPa, so should be the hardening moduli and the yield stress, and similarly for other dimensional parameters of the model.

10 Loadings

- Mechanical Loading
- Fatigue Loading
- Thermo-mechanical Loading
- Thermal Loading
- Electrical Loading

Mechanical Loading

Mechanical loadings are the most commonly used loadings in Digimat-MF. It covers different types of loadings, either stress or strain specified, which can be used to mimic loadings that could be applied to a material sample in a test laboratory. It goes from any kind of static tests, like pressure tests on closed systems, to dynamic tests. That means you can define the working strain rate and account for viscous effects, or simply make static load tests. The steady-state response of a material under harmonic loading is also available.

When performing a mechanical analysis in Digimat-MF, the assumption of isothermal analysis is made. To perform a temperature-dependent analysis, the user should select the thermo-mechanical analysis type.

The complete definition of a mechanical loading is done in 2 steps, first by selecting the loading type and the time history that should be applied to the material point and, then, by specifying the magnitude of the loading. Figure 10-1 illustrates the **Mechanical loading** tab.

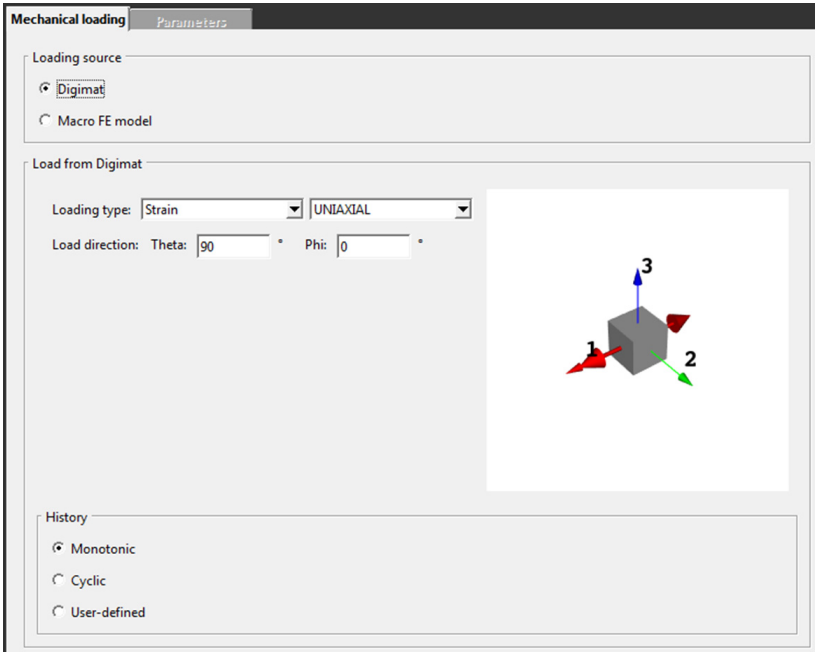


Figure 10-1 Mechanical loading menu.

In Digimat-MF, the boundary conditions that can be applied to the material point or representative volume element (RVE) under study can be of four types. Either can the user specify strain, stress, harmonic strain or harmonic stress components. These boundary conditions are always applied at the macroscopic level, i.e., at the composite level. It is not possible to apply boundary conditions at the phase level.

1st type: Strain loading

For each type of strain loading, there is a notion of **Initial strain** and **Peak strain** that the user must define. During the computation, the macro strains (11, 22, 12, ...) specified by the user are enforced by Digimat-MF on the boundaries of the RVE. The run terminates when the peak strain is reached.

To account for rate effects in the material response, a loading rate can also be specified. By default, the loading is set to the option **Use quasi-static loading**. The user should switch to **Define loading strain rate** and specify the strain value to account for viscous effect in the material response. This is to be used only with viscoelastic, elasto-viscoplastic and viscoelastic-viscoplastic materials.

Without Using the Finite Strain Option

Several strain loading types are available in Digimat-MF. Each type corresponds to a given macroscopic loading state. For instance, to reproduce a uniaxial tensile test, the **UNIAXIAL_1** strain loading type will be retained.

The different available strain loading types are presented hereafter.

- **UNIAXIAL_1**: the component 11 of the macroscopic strain is driven by the user. The other strain components are computed such that a macroscopic uniaxial stress state in the 1-direction is imposed. An example of **UNIAXIAL_1** loading is shown in the Figure 10-2.

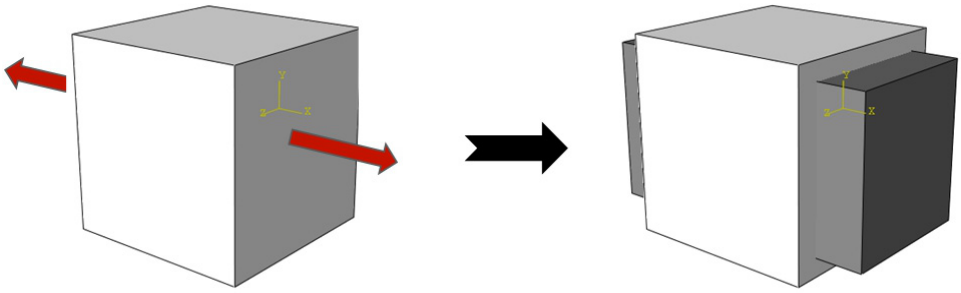


Figure 10-2 Illustration of the UNIAXIAL_1 loading.

A Load Direction can be assigned to **UNIAXIAL_1** Loading Type.

- **BIAXIAL1_2**: the component 11 and 22 of the macroscopic strain are driven by the user. The other strain components are computed such that a macroscopic biaxial stress state in the 1- and 2-directions is imposed.

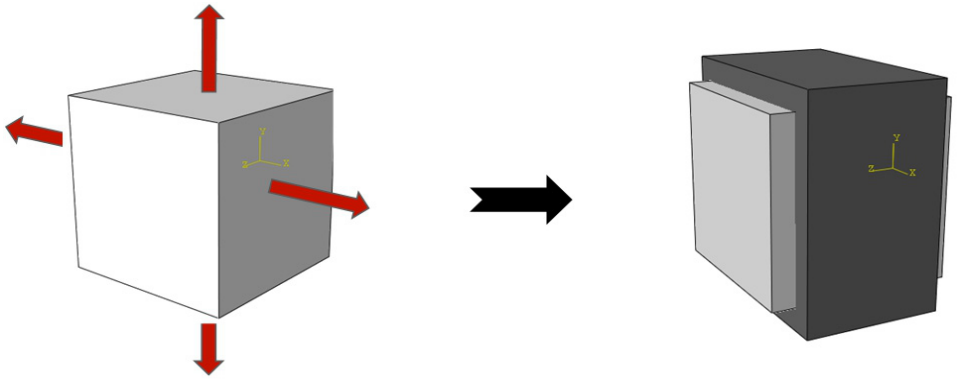


Figure 10-3 Illustration of the BIAXIAL1_2 loading.

- **SHEAR_12**: the component 12 of the macroscopic strain is driven by the user. The other strain components are computed such that a macroscopic simple shear stress state in the (1,2)-plane is imposed.

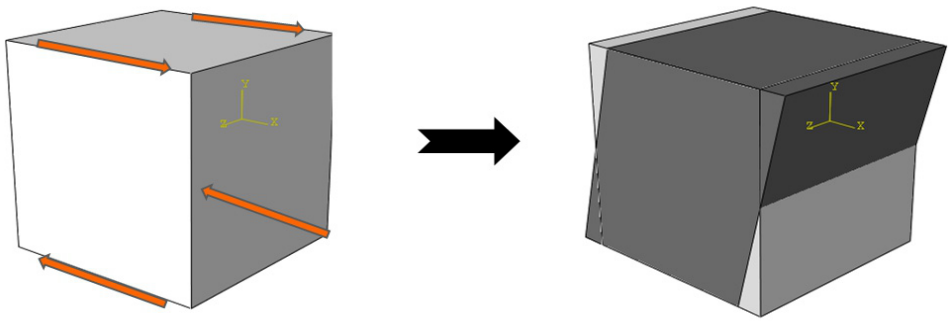
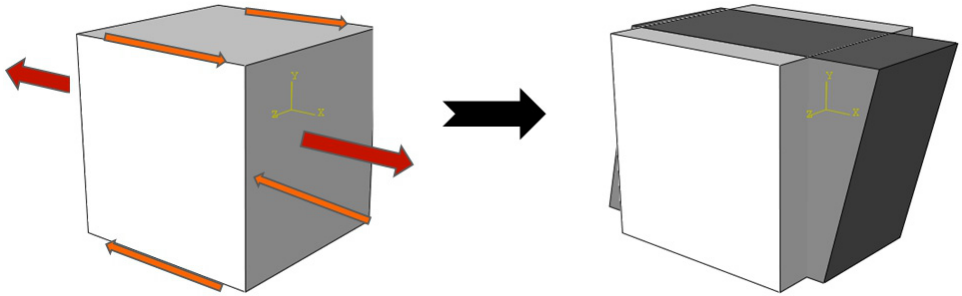


Figure 10-4 Illustration of the SHEAR_12 loading.

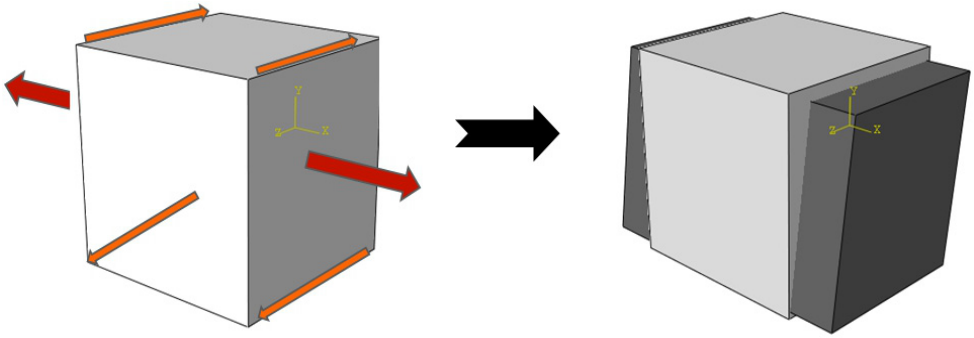
A Load Direction can be assigned to **SHEAR_12** Loading Type.

- **BIAXIAL1_12**: the component 11 and 12 of the macroscopic strain are driven by the user. The other strain components are computed such that a combination of a macroscopic uniaxial stress state in the 1-direction and a macroscopic simple shear stress state in the (1,2)-plane is imposed.

An Illustration of the BIAXIAL1_12 loading is shown in the following figure.



- **BIAXIAL1_23**: the component 11 and 23 of the macroscopic strain are driven by the user. The other strain components are computed such that a combination of a macroscopic uniaxial stress state in the 1-direction and a macroscopic simple shear stress state in the (2,3)-plane is imposed. An Illustration of the BIAXIAL1_12 loading is shown in the following figure.



- **GENERAL_2D**: the component 11, 22 and 12 of the macroscopic strain are driven by the user. The other strain components are computed such that a combination of a macroscopic 2D stress state is imposed in the (1,2)-plane.
- **GENERAL_3D**: a macroscopic 3D strain state is imposed on the boundaries of the RVE. The user should specify six strain components.
- **RELAXATION**: a macroscopic uniaxial stress state is imposed in the 1-direction of the RVE in two steps.

  In the first step, the component 11 of the macroscopic strain increases until the peak strain is reached. The number of increment to reach the peak strain has be to defined by the user; the default value is 10. In the second step, the component 11 of the macroscopic strain remains constant until the final time of the analysis is reached.

  A Load Direction can be assigned to **RELAXATION** Loading Type.

Using the Finite Strain Option:

In the case of **Finite strain** analyses, i.e., with hyperelastic or Leonov- EGP materials, a deformation gradient is applied rather than a strain field.

- **UNIAXIAL_1**: a uniaxial macroscopic deformation gradient state is imposed in the 1-direction, while the other components are automatically computed to achieve a uniaxial macroscopic stress state. A user defined uniaxial loading direction can be specified through the Theta and Phi parameters from the **Load direction** menu of the **Mechanical loading** tab (see Figure 10-7). In that case, the output fields will be thrown into the loading axis system, so the index 1 will always refer to the loading direction. The default **UNIAXIAL_1** loading direction corresponds to the configuration Theta= 90^0 and Phi= 0^0.
- **BIAXIAL1_2**: a biaxial macroscopic deformation gradient state is imposed in the 1- and 2-directions, while the other components are computed in order to achieve a macroscopic plane stress state.
- **SHEAR_12**: the following macroscopic deformation gradient is imposed on the boundaries of the RVE,

$$F = 1 + \gamma e_1 \otimes \qquad (10\text{-}1)$$

where $\mathbf{1}$ denotes the identity tensor, e_2 is the normal to the slip plane and e_1 is the slip direction in the slip plane. Digimat-MF computes a shear stress state in the (1,2)-plane and the following components of the Cauchy stress tensor: s_{11}, s_{22} and s_{33}. Unlike the infinitesimal strain case, these three components of the Cauchy stress tensor do not vanish in general.

- **BIAXIAL1_12**: a biaxial macroscopic deformation gradient resulting from a combination of the 11 and 12 components of the macroscopic deformation gradient is imposed. Digimat-MF computes a macroscopic stress state which is a combination of a biaxial macro plane stress state and a shear stress state in the (1,2)-plane.
- **BIAXIAL1_23**: a biaxial macroscopic deformation gradient resulting from a combination of the 11 and 23 components of the macroscopic deformation gradient is imposed. Digimat-MF computes a macroscopic stress state which is a combination of a biaxial macro plane stress state and a shear stress state in the (2,3)-plane.
- **RELAXATION**: a uniaxial macroscopic deformation gradient state is imposed in the 1-direction of the RVE in two steps; the other components are automatically computed to achieve a uniaxial macroscopic stress state. In the first step, the macroscopic deformation gradient increases until the peak strain is reached. The number of increment to reach the peak strain has be to defined by the user; the default value is 10. In the second step, the macroscopic deformation gradient remains constant until the analysis time is reached.

2nd Type: Stress

Four types of stress loadings are available:

- **UNIAXIAL_1**: a macroscopic uniaxial stress state in the 1-direction is imposed on the RVE.

  A Load Direction can be assigned to **UNIAXIAL_1** Loading Type.
- **HYDROSTATIC**: a macroscopic hydrostatic pressure stress state is imposed on the RVE.
- **TRIAXIAL1_2_3**: a macroscopic triaxial stress state is imposed on the RVE.
- **CREEP**: a macroscopic uniaxial stress state is imposed in the 1-direction of the RVE in two steps. In the first step, the stress increases until the peak stress is reached. The number of increment to reach the peak stress has be to defined by the user; the default value is 10. In the second step, the macroscopic stress remains constant until the analysis time is reached.

  A Load Direction can be assigned to **CREEP** Loading Type.

For each type of stress loading, an **Initial stress/pressure** and a **Peak stress/pressure** must be defined by the user. During the computation, the macro stresses (σ_{11}, σ_{22}, σ_{12} ,...) are enforced on the boundaries of the RVE by Digimat-MF. The run terminates when the peak value is reached.

3rd Type: Harmonic Strain

Seven types of harmonic strain loadings are available: UNIAXIAL, BIAXIAL1_2, BIAXIAL1_12, BIAXIAL1_23, SHEAR_12, GENERAL_2D and GENERAL_3D. These loadings are the harmonic equivalent of the strain loading types available described hereinbefore. In the harmonic case, the user is no more asked for Initial strain nor Peak strain but for a complex value definition of each harmonic strain component.

These complex values can be input as **Magnitude and Phase** or as **Real and Imaginary**, according to the user preference, see Figure 10-5). If the **Magnitude and Phase** input method is selected, the phase unit is the degree.

The harmonic loading must be completed by the definition of the oscillation frequencies at which a result is desired. Frequencies are determined through a sampling procedure based on the **Lowest frequency**, the **Highest frequency** and the **Number of frequencies**. **Logarithmic** or **Uniform** sampling can be performed. The lowest and highest frequencies are always included in the computed sample.

During the computation, the macro strains (11, 22, 12,...) specified by the user are enforced by Digimat- MF on the boundaries of the RVE for each specified frequency. The run terminates when the complex values of each component are reached.

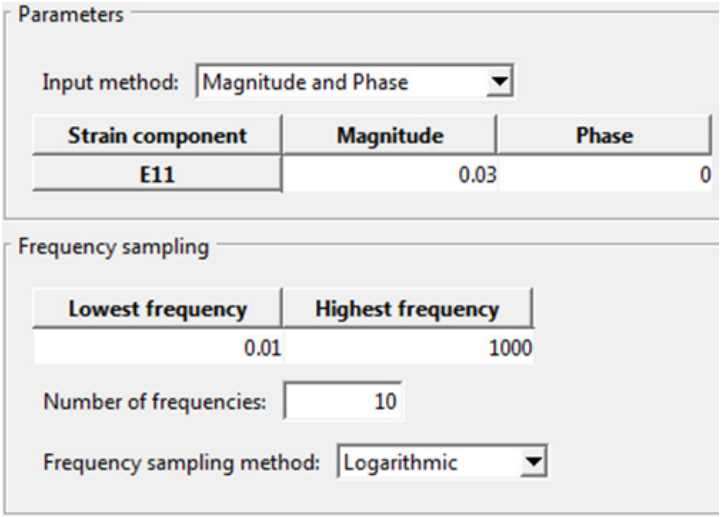


Figure 10-5 Illustration of the user defined UNIAXIAL_1 loading parameter tab (harmonic strain case).

4th type - Harmonic Stress

Three types of harmonic stress loadings are available: UNIAXIAL, HYDROSTATIC and TRIAXIAL1_2_3. These loadings are the harmonic equivalent of the stress loading types available described herein before. In the harmonic case, the user is no more asked for Initial stress/pressure nor Peak stress/pressure but for a complex value definition of each harmonic stress component. These complex values can be input as **Magnitude and Phase** or as **Real and Imaginary**, according to the user preference, see Figure 10-6). If the **Magnitude and Phase** input method is selected, the phase unit is the degree.

The harmonic loading must be completed by the definition of the oscillation frequencies at which a result is desired. Frequencies are determined through a sampling procedure based on the **Lowest frequency**, the **Highest frequency** and the **Number of frequencies**. **Logarithmic** or **Uniform** sampling can be performed. The lowest and highest frequencies are always included in the computed sample.

During the computation, the macro stresses (σ_{11}, σ_{22}, σ_{12},...) are enforced on the boundaries of the RVE by Digimat-MF for each specified frequency. The run terminates when the complex values of each component are reached.

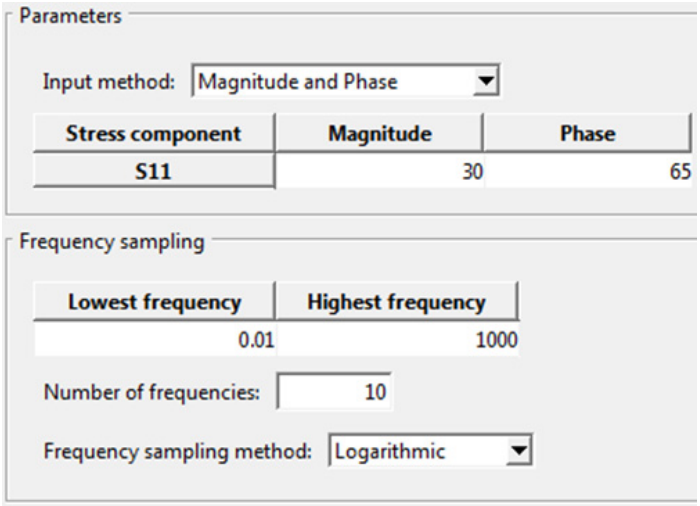


Figure 10-6 Illustration of the user defined UNIAXIAL_1 loading parameter tab (harmonic stress case).

Load Direction

A user defined uniaxial loading direction can be specified through the Theta and Phi parameters from the **Load direction** menu of the **Mechanical loading** tab.

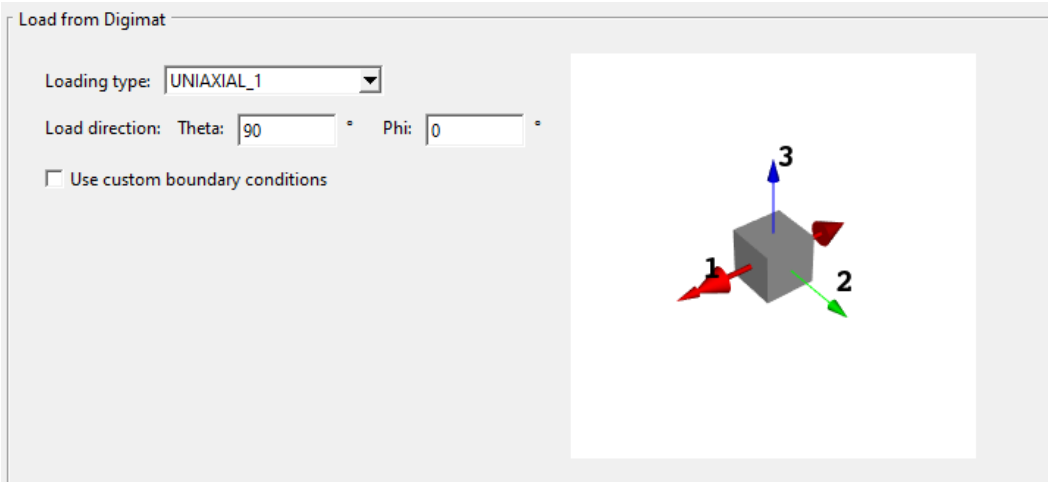


Figure 10-7 Illustration of the user defined UNIAXIAL_1 loading (strain case).

The above image shows the illustration of the user defined **UNIAXIAL_1** loading (strain case). In that case, the output fields will be thrown into the loading axis system, so the index 1 will always refer to the loading direction.

The default **UNIAXIAL_1** loading direction corresponds to the configuration Theta= 90^0 and Phi= 0^0.

Loading History Types

For non-harmonic loadings, in addition to specifying the loading type, the user should specify the history type the loading follows. The history type defines the time factor f (t) that scales the loading:

$$\mathcal{L}(\sigma|\varepsilon, t) = f(t)L(\sigma|\varepsilon) \tag{10-2}$$

Digimat-MF offers three types of history loading for mechanical loading: monotonic, cyclic and user-defined.

Monotonic

A ramp loading is applied from the initial loading value to the peak value specified for each loading component.

Cyclic

Cyclic loadings consist of successive loading/unloading from the initial loading value to its peak and negative peak value (see Figure 10-8). Four parameters are available for cyclic loading:

- Initial value
- Peak value
- Number of cycles
- Load ratio

The cyclic load will go first from the initial value to the peak value, then it will oscillates between the peak value and the minimum value until the number of cycles is reached. The minimum value is computed as the peak value * load ratio.

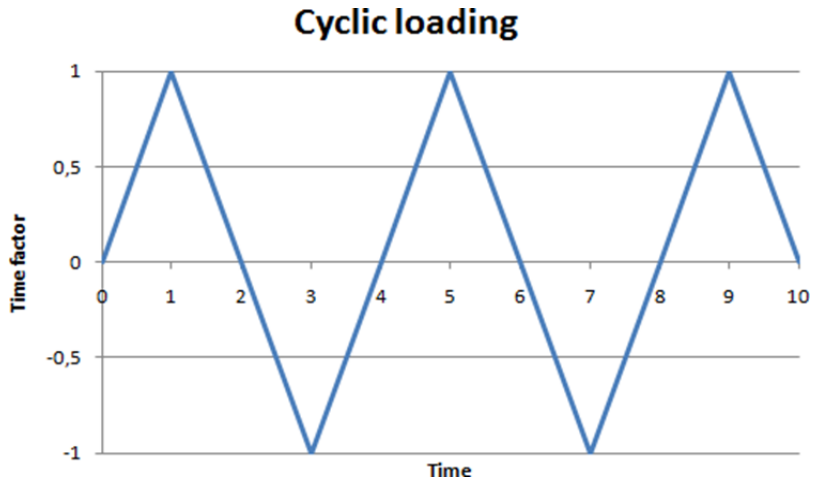


Figure 10-8 Illustration of the cyclic loading. Time factor versus time.

User-defined Loading Scheme

The user-defined loading scheme allows the user to specify specific time factor f (t) that will scale the loading as the simulation goes on. This very general history type allows the user to apply complex loadings on the boundaries of the RVE. This type of loading history is illustrated in the Example section.

To define a user-defined loading scheme, a function (see section Functions) should be created and assigned to the loading type. This function can be applied relatively or absolutely, i.e., to scale or override the defined loading peak values.

This loading scheme offers a wide range of loading possibilities. For instance, cyclic loadings with different strain rates for each cycle can be defined, as well as loadings with different peak values in different directions having their peak values reached at different time instants.

For the **UNIAXIAL_1** strain loading type, the time factor is applied on the 11-component of the macroscopic strain tensor, $\varepsilon_{11}(t)$. As for a monotonic and a cyclic loading, the 22- and the 33-components of the macroscopic strain, $\varepsilon_{22}(t)$ and $\varepsilon_{33}(t)$, are computed by solving the following set of nonlinear equations, which approximates the stress dependency on the strain state; see Doghri, (2000) for more details.

$$\begin{cases} \sigma_{22}(\varepsilon_{22}, \varepsilon_{33}) = 0, \\ \sigma_{33}(\varepsilon_{22}, \varepsilon_{33}) = 0 \end{cases} \tag{10-3}$$

For the **SHEAR_12** strain loading type, the time factor is applied on the 12-component of the macroscopic strain, $\varepsilon_{12}(t)$.

For biaxial loadings, the time factor is applied on each specified component of the macroscopic strain tensor. Different time factor can be applied to the strain tensor components. In this case, the ratio between the strain tensor components is not constant as the loading is applied.

Note that for each x-y function, the initial and final times must be the same. Also, the initial strain/stress/temperature will be computed directly from the loading function at time t = 0.

For the **GENERAL_2D** strain loading type, a time factor should be applied to each component (1,2)- plane component of the macroscopic strain tensor, $\varepsilon_{11}(t)$, $\varepsilon_{22}(t)$ and $\varepsilon_{12}(t)$. The other components of the macroscopic strain tensor are computed by solving the following set of nonlinear equations, which enforces the plane stress state,

$$\begin{cases} \sigma_{13}(\varepsilon_{13}, \varepsilon_{23}, \varepsilon_{33}) = 0 \\ \sigma_{23}(\varepsilon_{13}, \varepsilon_{23}, \varepsilon_{33}) = 0 \\ \sigma_{33}(\varepsilon_{13}, \varepsilon_{23}, \varepsilon_{33}) = 0 \end{cases} \tag{10-4}$$

For the **GENERAL_3D** strain loading type, a time factor should be applied to all six components of the macroscopic strain tensor.

Fatigue Loading

A fatigue loading aims at computing in Digimat-MF an S-N curve consistent with the one obtained from an experimental test campaign on material specimens. This S-N curve can be computed for a predefined range of numbers of cycles or stress amplitudes (see here under). For any fatigue analysis performed in Digimat-MF, the assumption of isothermal analysis is made.

The only boundary condition that can be applied to the material point or representative volume element (RVE) under study is a **UNIAXIAL stress loading**. This means that a macroscopic uniaxial stress state in a given direction is imposed on the RVE.

Numbers of Cycles

The first way to compute a macroscopic S-N curve with Digimat-MF is by defining a target range of numbers of cycles (see Figure 10-9), i.e., **minimum** and **maximum** strictly positive values. It is representative for the S-N curve computation performed in the framework of Digimat-CAE fatigue interfaces (see section Coupling to Fatigue).

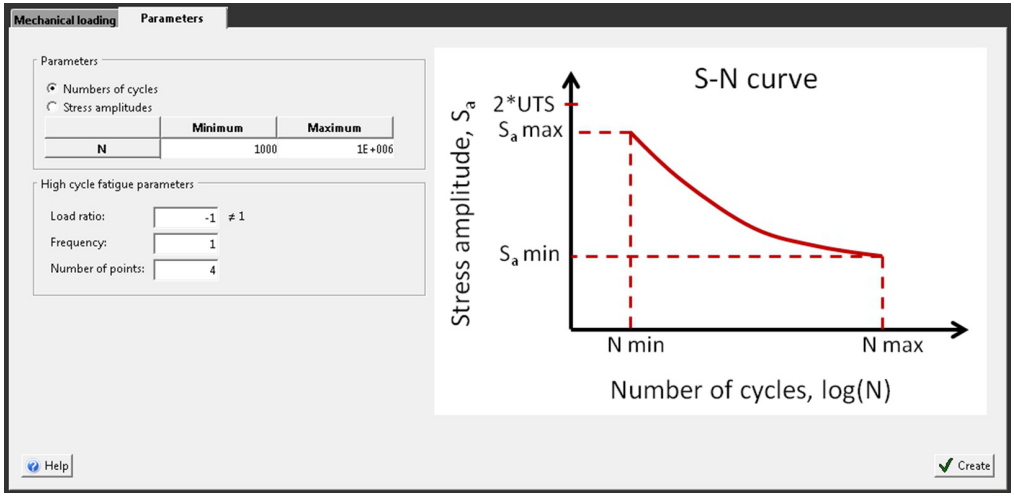


Figure 10-9 Number of cycles loading definition.

Several additional parameters further define the targeted S-N curve:

- the **load ratio**, i.e., the ratio between minimum and maximum stresses imposed during a corresponding real cyclic test, if the fatigue failure indicator includes a mean stress sensitivity definition (see Mean Stress Sensitivity Definition) or if the modified Gerber model is selected
- the **frequency** if the analysis involves a viscoelastic matrix
- the **number of points** evenly distributed – on a log scale – between the minimum and maximum numbers of cycles

For each point, the stress amplitude is computed according to the fatigue failure indicator (see Fatigue Models and Modified Gerber Model). The result is an S-N curve for a given microstructure.

Note that for the modified Gerber model, this loading is supported only in association with stress based Tsai-Hill 3D TI failure criterion (defined at PG level).

Stress Amplitudes

The second way to compute a macroscopic S-N curve with Digimat-MF is by defining a target range of stress amplitudes (see Figure 10-10), i.e., **minimum** and **maximum** strictly positive values. Combined to a fatigue model (see Fatigue Models), it is representative for the number of cycles computation performed in the framework of Digimat-CAE implicit FE interfaces (see High Cycle Fatigue).

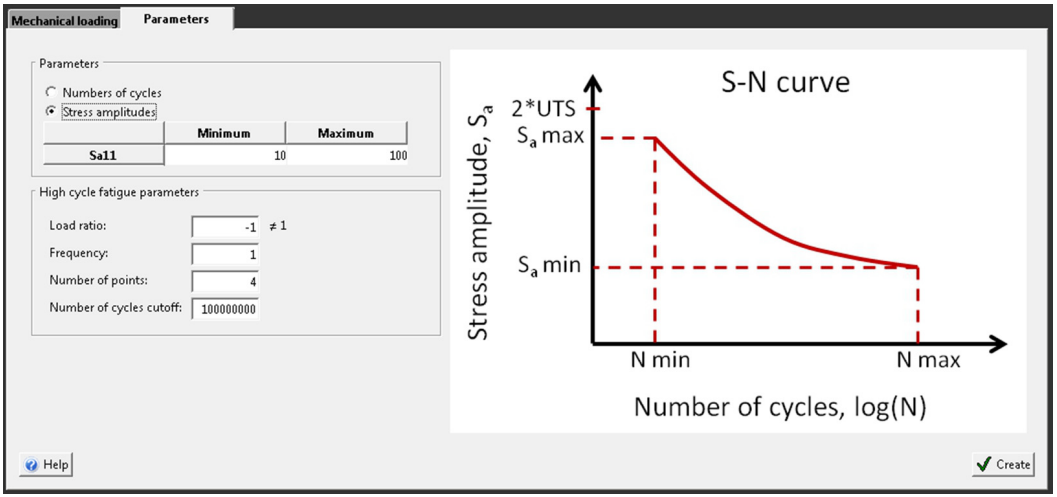


Figure 10-10 Stress amplitude loading definition.

The additional parameters defining the targeted S-N curve mostly correspond to the number of cycles loading definition.

- The **load ratio** and **frequency** are not available with the matrix damage model (see Matrix Damage Fatigue Model). On the one hand, this model does not involve the mean stress sensitivity definition strongly related to the load ratio input. On the other hand, it involves a high cycle damage matrix, i.e., not viscoelastic.
- The **number of points** evenly distributed – on a linear scale – between the minimum and maximum stress amplitudes may not be strictly obtained. Indeed, if the number of cycles computed for some stress amplitude becomes larger than the **number of cycles cutoff**, the analysis stops and smaller stress amplitudes are not considered.

For the matrix damage fatigue model, a load ratio $R = \sigma_{min}/\sigma_{max} = -1$ is applied on the boundary of the RVE when cyclic loading is selected. The corresponding loading cycle is constructed automatically by Digimat-MF. The corresponding evolution of the macroscopic 11-component of the Cauchy stress tensor is presented in Figure 10-11.

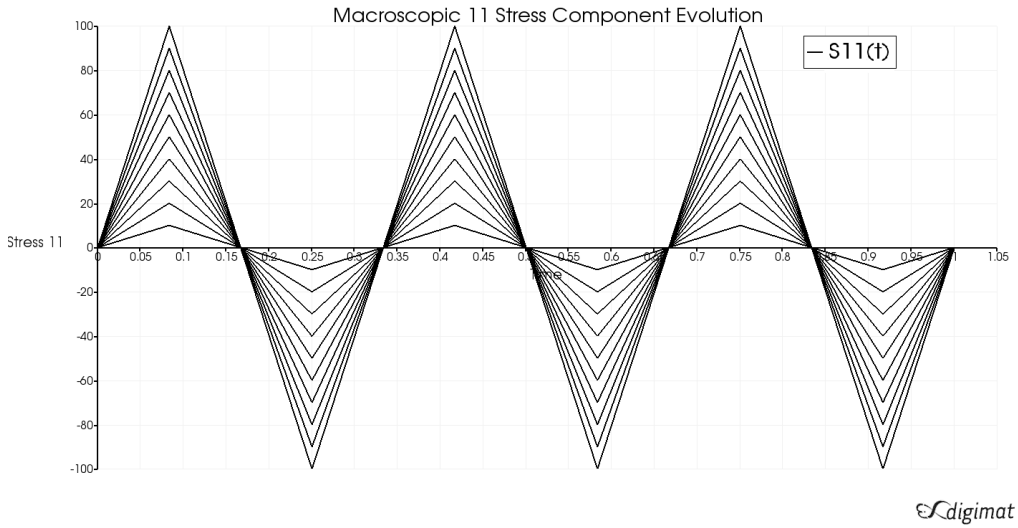


Figure 10-11 Time evolution of the 11-component of the Cauchy stress tensor.

To compute a macroscopic S-N curve corresponding to a load ratio R different from -1, select the user-defined history loading in the mechanical loading tab (see Figure 10-1). The unit loading cycle is then described by a user-defined history function, which gives the evolution of the macroscopic 11-component of the Cauchy stress tensor over time. This unit cycle is then multiplied by a factor ranging between **minimum** and **maximum** values (see Figure 10-12).

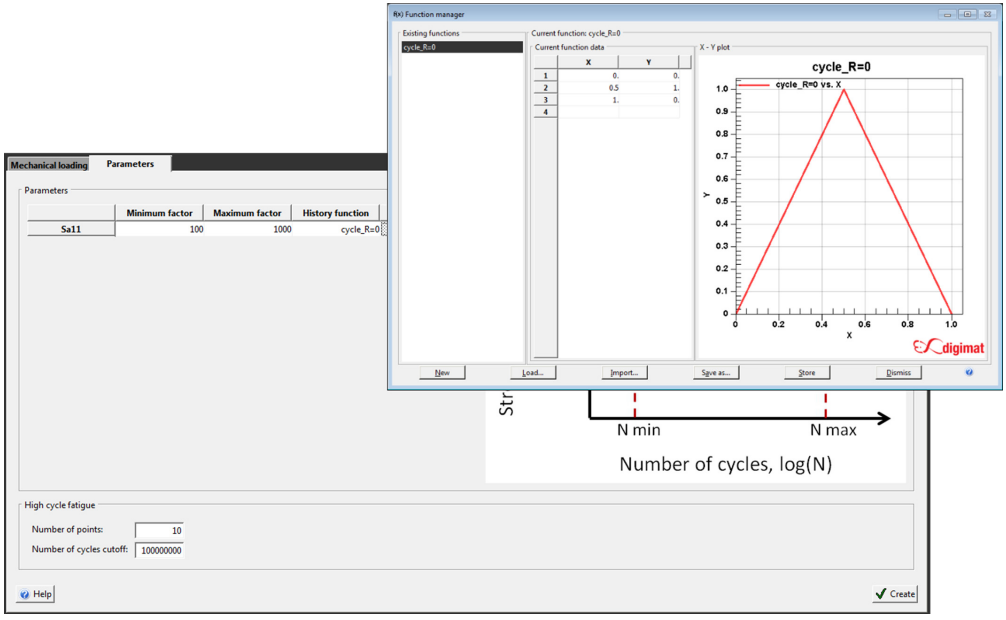


Figure 10-12 Stress amplitude loading definition with user-defined load cycle.

Thermo-mechanical Loading

Digimat-MF not only offers mechanical analyses but also thermo-mechanical ones. In such an analysis, part of the material deformation is assumed to come from a temperature variation as the loading is applied on the RVE. Such type of analysis is typically performed to evaluate the residual stresses that build up during the cool down of an injected plastic part.

The definition of a thermo-mechanical loading in Digimat-MF is a two step process. The user first defines the mechanical loading (see section Mechanical Loading) and then the thermal one. This section describes the definition of the temperature variation over the loading duration.

Figure 10-13 Thermal loading tab in the graphical user interface.

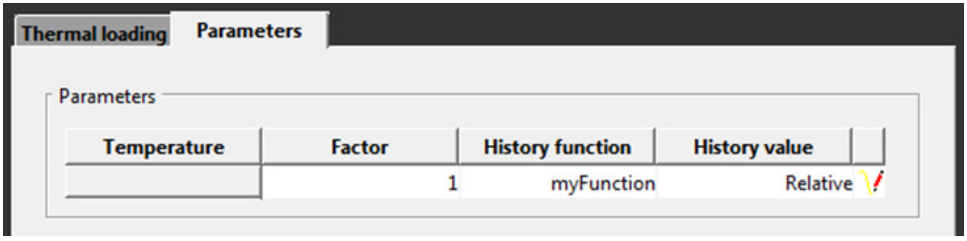


Figure 10-14 Parameters tab in the graphical user interface.

Loading History Types

Digimat-MF offers three types of history loading for thermal loading: monotonic, cyclic and user-defined. The loading history defines how the loading evolves with time as the simulation is being run. It can be combined with any type of mechanical history loading. The number of parameters to input depends on the selected option, as described below.

The history type defines the time factor f (t) that scales the loading

$$\mathcal{L}(\sigma|\varepsilon, t) = f(t)L(\sigma|\varepsilon) \tag{10-5}$$

Monotonic

A ramp loading is applied from the initial temperature to the peak temperature.

Cyclic

Cyclic loadings consist of successive loading/unloading from the initial temperature to the peak temperature and the negative peak temperature (see Figure 10-15).

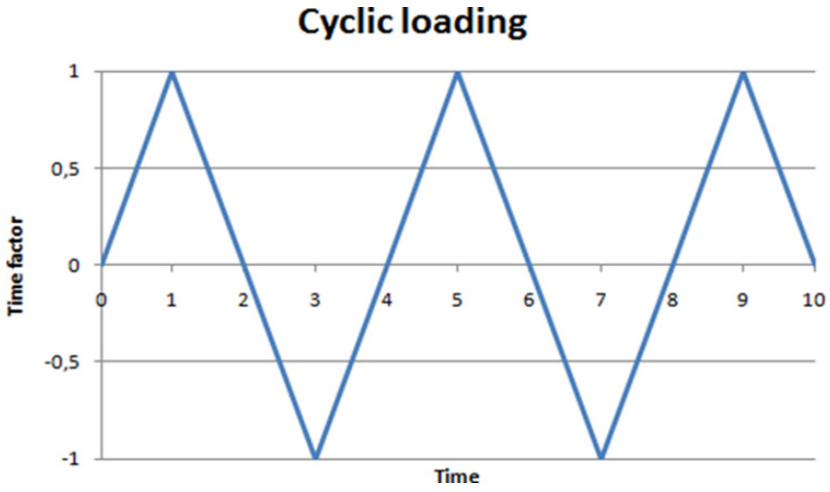


Figure 10-15 Illustration of the cyclic loading. Time factor versus time.

User-defined Loading Scheme

The user-defined loading scheme allows the user to specify a specific time factor f (t) that will scale the loading as the simulation goes on. This very general history type allows the user to apply complex loadings on the RVE. This type of loading history is illustrated in the example section and in Figure 10-13 and Figure 10-14.

To define a user-defined loading scheme, a function (see Functions) should be created and assigned to the thermal loading. This function can be applied relatively or absolutely, i.e., to scale or override the defined temperature initial and peak values.

This loading scheme offers a wide range of loading possibilities. For instance, cyclic loadings with different rates for each cycle can be defined, as well as loadings with different peak values.

Thermal Loading

Different types of analyses can be carried out with Digimat-MF. Among them thermal ones, during which homogenization is performed on the thermal conductivity of the composite.

This section presents the definition of thermal loadings for Digimat-MF analyses, which is a two-step process:

1. Define the history loading.
2. Define the loading parameters.

A Load Direction can be assigned.

Loading History

Three loading histories are available in Digimat: **Monotonic, Cyclic, User-defined**. These are functions f (t) that scale the default loading with respect to time

$$\mathcal{L}(T, t) = f(t)L(T) \tag{10-6}$$

Monotonic

A ramp loading is applied from the initial loading value to the peak value specified for each loading component.

Cyclic

Cyclic loadings consist of successive loading/unloading from the initial loading value to its peak and negative peak value (see Figure 10-16).

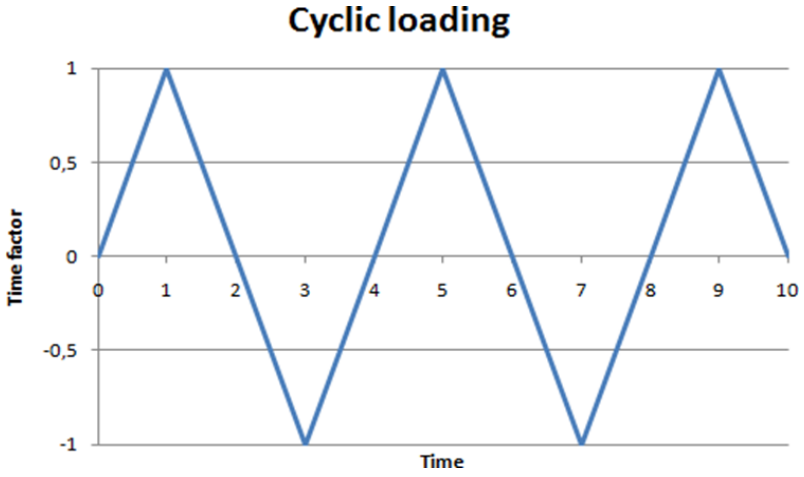


Figure 10-16 Illustration of the cyclic loading. Time factor versus time.

User-defined Loading Scheme

The user-defined loading scheme allows the user to specify specific time factor f(t) that will scale the loading as the simulation goes on. This very general history type allows the user to apply complex loadings on the boundaries of the RVE.

To define a user-defined loading scheme, a function (see section Functions in *Getting Started Guide*) should be created and assigned to the loading type. This function can be applied relatively or absolutely, i.e., to scale or override the defined loading peak values.

This loading scheme offers a wide range of loading possibilities. For instance, cyclic loadings with different loading rates for each cycle can be defined.

Loading Parameters

For thermal analyses, a temperature gradient needs to be applied across the volume element. Such a loading is uniaxial along the 1-axis of the RVE. Its definition requires an initial and a peak values. These correspond to the gradient magnitudes at time zero and at peak time.

Electrical Loading

Different types of analyses can be carried out with Digimat-MF. Among them electrical ones, during which homogenization is performed on the electrical conductivity of the composite.

This section presents the definition of electrical loadings for Digimat-MF analyses, which is a two-step process:

1. Define the history loading.
2. Define the loading parameters.

A Load Direction can be assigned.

Loading History

Three loading histories are available in Digimat: **Monotonic, Cyclic, User-defined**. These are functions f (t) that scale the default loading with respect to time

$$\mathcal{L}(V,t) = f(t)L(V) \tag{10-7}$$

Monotonic

A ramp loading is applied from the initial loading value to the peak value specified for each loading component.

Cyclic

Cyclic loadings consist of successive loading/unloading from the initial loading value to its peak and negative peak value (see Figure 10-17).

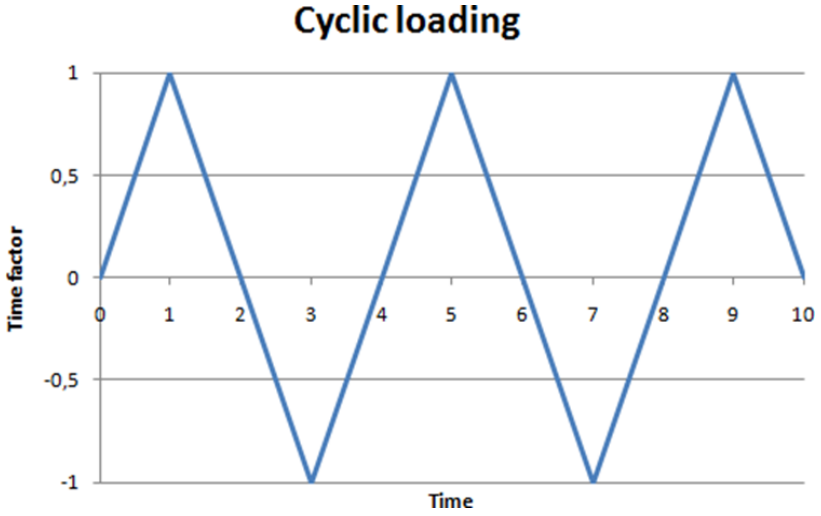


Figure 10-17 Illustration of the cyclic loading. Time factor versus time.

User-defined Loading Scheme

The user-defined loading scheme allows the user to specify specific time factor f (t) that will scale the loading as the simulation goes on. This very general history type allows the user to apply complex loadings on the boundaries of the RVE.

To define a user-defined loading scheme, a function (see Functions in *Getting Started Guide*) should be created and assigned to the loading type. This function can be applied relatively or absolutely, i.e., to scale or override the defined loading peak values.

This loading scheme offers a wide range of loading possibilities. For instance, cyclic loadings with different loading rates for each cycle can be defined.

Loading Parameters

For electrical analyses, a voltage gradient needs to be applied across the volume element. Such a loading is uniaxial along the 1-axis of the RVE. Its definition requires an initial and a peak values. These corresponds to the voltage gradient magnitudes at time zero and at peak time.

11 Results

- Output Management
- Output Files and Variables
- Plotting Tools
- Plotting Failure Envelopes
- Plotting Carpet Plots
- Plotting Forming Limit Diagrams
- Engineering Results

Output Management

Digimat offers to the user more flexibility and additional choices about which results to output: as presented on Figure 11-1, the user can manage:

- fields for each phase
- fields at the macroscopic level
- the engineering file
- the dependency elastic moduli file
- the fatigue results file(s) (if relevant)
- the log file

The detailed available fields for each section will be firstly presented. Secondly a presentation of the new output fields is done.

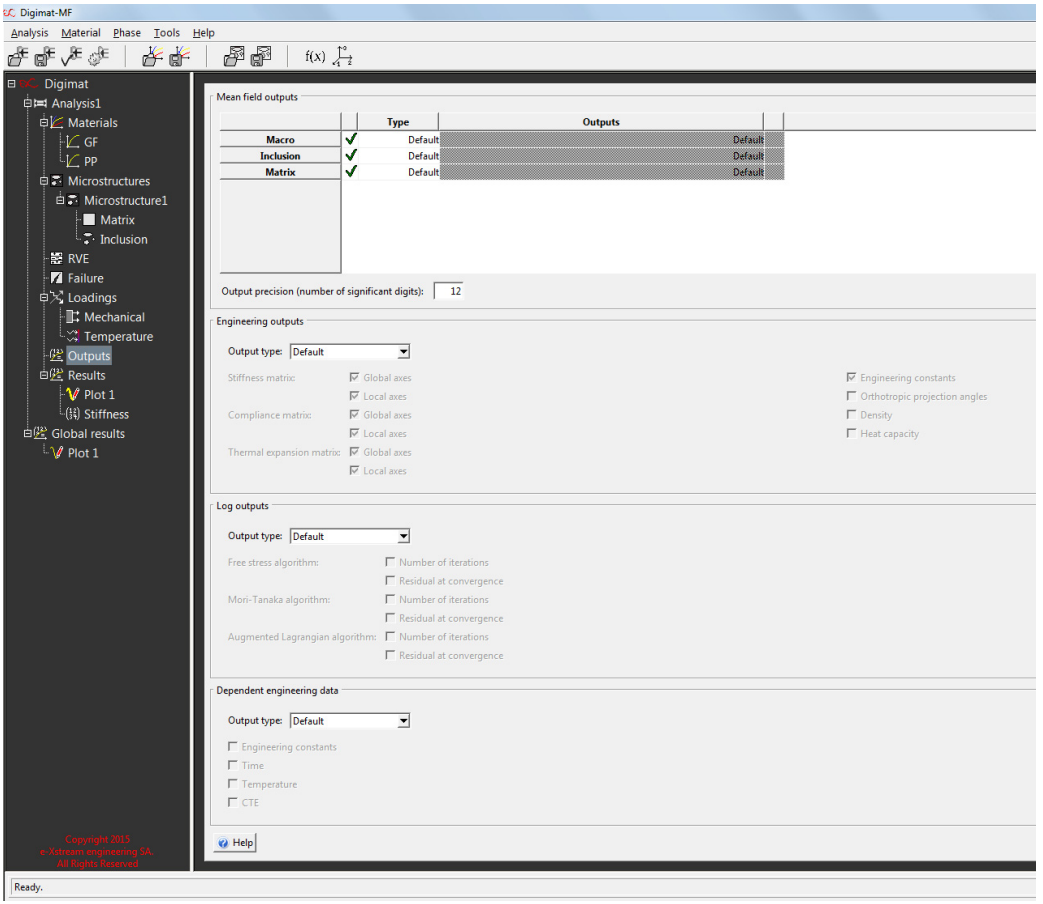


Figure 11-1 Output definition interface window.

Digimat also offers the possibility to the user to manage field outputs at several levels:

- RVE level, and
- phase level.

General Output Fields Available per Material Model

Under the **Output** section of the Digimat-MF GUI, a tab of the mean field output at the macroscopic level and at the phases level appears, as presented on Figure 11-2. From this section, the user can manage the output fields for each phase and the output field at the macroscopic level.

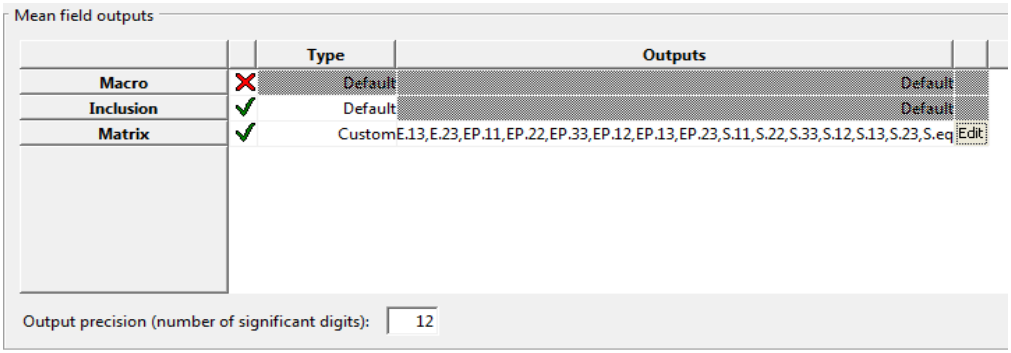


Figure 11-2 Mean Field macroscopic and per phase outputs.

The name of the phase defining the microstructure of the RVE is set to the right column. "Macro" refers to the RVE level. Three types of output are available:

- None: If a red cross is set near the name of the phase, no field will be set in output for this phase. For example, following Figure 11-2, the Macroscopic Result (`*.mac`) file won't be printed.
- Default: the default output fields will be printed.
- Custom: the user can chose which output fields will be printed.

For each kinds of materials, a summary of the default output fields to print in all the output files available in Digimat-MF are given below.

- (thermo-)Elastic model
  - Time of the analysis: time
  - Strain: E
  - Cauchy Stress: S
  - Volume variation: dVol (scalar)
  - Temperature field: Temp (scalar)

- (thermo-)Elasto-Plastic (J_2-Plasticity & Drucker-Prager) / (thermo-)Elasto-ViscoPlastic / Viscoelastic-Viscoplastic models
  - Time of the analysis: time
  - Strain: E
  - Plastic Strain: EP
  - Cauchy Stress: S
  - Volume variation: dVol (scalar)
  - Temperature field: Temp (scalar)

  NOTE: At RVE level, EP is not available
- Chaboche Cyclic plasticity model
  - Time of the analysis: time
  - Strain: E
  - Plastic Strain: EP
  - Cauchy Stress: S
  - Back stress: X
  - Volume variation: dVol (scalar)

  NOTE: At RVE level, EP and X are not available
- Lemaitre-Chaboche damage model
  - Time of the analysis: time
  - Strain: E
  - Plastic Strain: EP
  - Cauchy Stress: S
  - damage parameter: damD
  - Volume variation: dVol (scalar)

  NOTE: At RVE level, EP and damD are not available
- Hyperelastic / Thermo-Hyperelastic model
  - Time of the analysis: time
  - Deformation Gradient: F
  - Left Cauchy-Green strain tensor: LCG
  - Green-Lagrange strain: E
  - Nominal Strain: NE
  - Cauchy stress: S
  - Kirchhoff stress: KS
  - Nominal Stress: NS

- First invariant of the Left Cauchy-Green strain tensor: I1
- Second invariant of the Left Cauchy-Green strain tensor: I2
- Third invariant of the Left Cauchy-Green strain tensor: I3
- Maximal invariant: IMax
- Volume variation: dVol

- Leonov-EGP model
  - Time of the analysis: time
  - Green-Lagrange strain: E
  - Cauchy stress: S
  - Plastic Strain: EP
  - Hardening stress: Sh
  - Driving stress: Sd

  NOTE: At RVE level, EP, HS and DS are not available
- Ohm model
  - Electrical Flux: J
  - Voltage gradient: Vgrad
- Fourier model
  - Heat flux: Hflux
  - Thermal gradient: Tgrad

Particular Outputs for Failure Indicators

Additional fields are available when failure criteria are assigned, depending on the level of assignment, on the type of failure mechanism (standard/FPGF/Progressive failure), and on some options triggered in the Failure assignment tab (see section Failure Criterion Definition & Assignment). These fields are available by default or can be customized by the user as presented on Figure 11-3.

Output for Standard Failure Indicators

If one or more failure indicator is associated to one or more phase of the composite or at the macroscopic level, it is possible to output them or not (see Figure 11-3). By default, failure criteria are printed in the corresponding output phase. If more than one failure indicator is required at the macroscopic level or at a phase level, all failure indicator will be printed.

It is not possible to choose a particular failure indicator among all the output data.

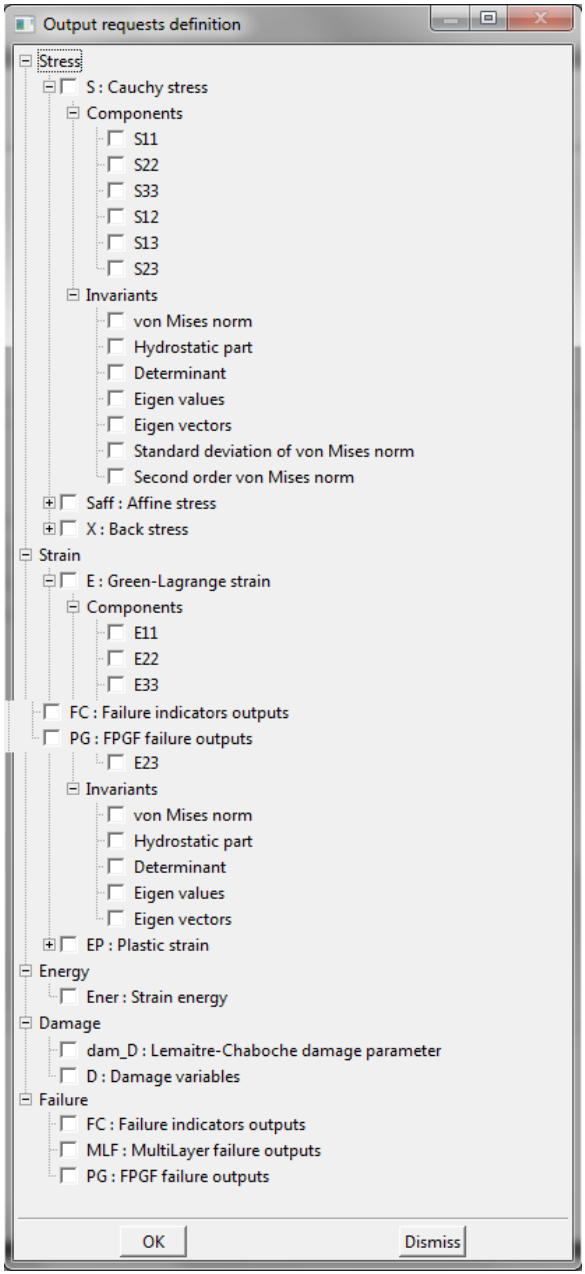


Figure 11-3 Output request definition.

Output for Progressive Failure Indicators

Remark: The output FC has a different meaning for Standard and Progressive failure indicators. Progressive failure indicators are computed on the basis of the effective (undamaged) stress tensor SEff, instead of the apparent (damaged) stress tensor S which is used for Standard or FPGF failure.

Moreover, if at least one progressive failure mechanism is defined at some Phase or Composite level, two new outputs will be available in the phase output data:

- Linear elastic model with Progressive Failure mechanism (see subsection Theory)
  - Damage variables: D
  - Effective stress (and its invariants): SEff

These outputs are available in the corresponding Phase or Macroscopic file. The damage variables output is activated by default, but not the effective stresses.

Output for FPGF

If FPGF is associated to one phase of the composite or at the macroscopic level, it is possible to output them or not. By default, FPGF data are printed in the corresponding output files as well as the associated failure indicator.

Output for Multilayer Failure

If the **multilayer failure** controls are applicable (see subsection Failure for Multilayer RVE), two additional outputs are available at the macroscopic level (independently of the assignment level and of the failure mechanism):

- FI_AVG is the average value of the failure indicator
- FI_THK is the thickness fraction of failed layers

Other Particular Output Fields

New fields and particular fields are available depending firstly on the material model and secondly, on the used homogenization procedure. These fields are available by default or can be customized by the user as presented on Figure 11-3.

Particular Output Fields for Finite Strain Analysis

For analysis involving hyperelastic materials and/or Leonov model, the microstructure of the RVE can evolve during the loading path. The user can follow the microstructure's evolution through the inclusion's shape evolution and the inclusion's orientation evolution:

- inclusion's shape evolution: Inclusion_Shape
- inclusion's orientation evolution: Inclusion_Orientation

Note that these fields are only available at the inclusion level. By default they are set in output.

Particular and Tensor Fields

- Second order homogenization method

  For the second order homogenization scheme, the standard deviation and the mean square field of the Cauchy stress tensor, S, are required (see Figure 11-3):

  - Standard deviation of the von Mises norm
  - Second order von Mises norm

  These are only available at the phase level in which an elasto-plastic or elasto-viscoplastic material is associated.

- Discrete affine method

  Only if the discrete affine method is used, the affine stress Saff can be written in the output file, but only at the phase level where an (thermo-)elasto-viscoplastic materials is defined.

- Tensor fields

  In Digimat-MF the user has the possibility to print only one or many components of a particular field and some invariant of these fields (see Figure 11-3). For example, for the Cauchy stress (symmetric second order tensor) S, and the deformation gradient (non symmetric second order tensor) F, it is possible to print the following particular components:

  - Component:

    S11 / S22 / S33 / S12 / S13 / S23: Symmetric tensor

    F11 / F21 / F31 / F12 / F22 / F32 / F31 / F32 / F33: Non Symmetric tensor

  - Invariants:
    - Von Mises norm (second invariant)

      $$X_{eq} = \sqrt{\frac{3}{2}\mathrm{dev}X:\mathrm{dev}X} \text{ for field } X \qquad (11\text{-}1)$$

    - Hydrostatic part (first invariant):

      $$X_m = \frac{1}{3}\mathrm{Tr}(X) \text{ for field } X \qquad (11\text{-}2)$$

    - Determinant (third invariant)
    - Eigenvalues
    - Eigenvectors

Output Available in Engineering File (.eng)

The engineering file contains all information about the macroscopic stiffness and compliance and the engineering constants, if available. Three types of output are available:

- None: The Engineering Moduli `*.eng` file won't be created.
- Default: the default output fields will be printed.
- Custom: the user can chose which fields to output.

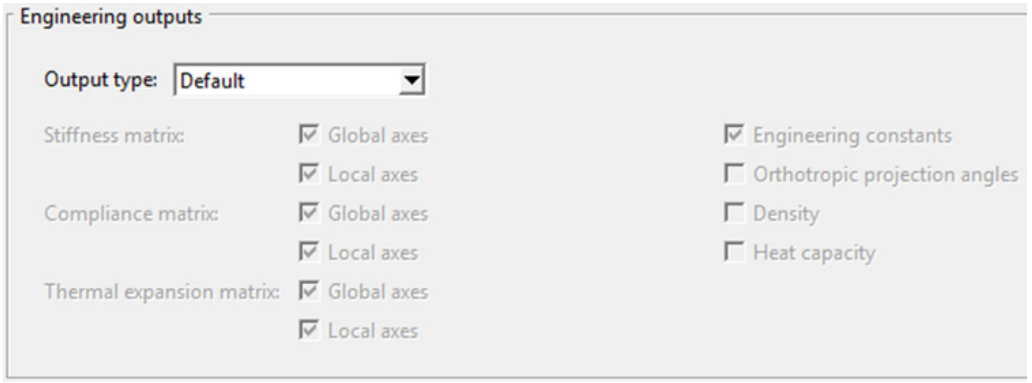


Figure 11-4 Output available for engineering file.

- time: Time of the analysis
- temp: Temperature field
- C_{loc} stands for:
  - local stiffness matrix for (thermo-)mechanical analysis
  - local thermal conductivity for thermal conductivity
  - local electrical conductivity for electrical conductivity
- C_{glob} stands for:
  - global stiffness matrix for (thermo-)mechanical analysis
  - global thermal conductivity for thermal conductivity
  - global electrical conductivity for electrical conductivity
- S_{loc} stands for:
  - local compliance matrix for (thermo-)mechanical analysis
  - local thermal resistivity for thermal conductivity
  - local electrical resistivity for electrical conductivity
- S_{glob} stands for:
  - global compliance matrix for (thermo-)mechanical analysis
  - global thermal resistivity for thermal conductivity
  - global electrical resistivity for electrical conductivity
- AlphaLoc: Local expansion matrix (not printed for thermo-viscoelastic materials)

- AlphaGlob: Global expansion matrix (not printed for thermo-viscoelastic materials)
- EngModuli: Engineering constants (orthotropic or transversely isotropic projection)
- OrthoAngles: Orthotropic angles (only for an orthotropic projection)
- Density: Global density of the composite
- HeatCapacity: Global heat capacity of the composite

Output Available in Dependency Elastic Moduli File (.dem)

This file contains the evolution of the macroscopic engineering moduli and the macroscopic thermal coefficient as a function of temperature for thermo-mechanical analysis, and as a function of the time for viscoelastic composite. Three types of output are available:

- None: the Dependency Elastic Moduli (`*.dem`) file won't be created.
- Default: the default output fields will be printed.
- Custom: the user can chose which fields to output.

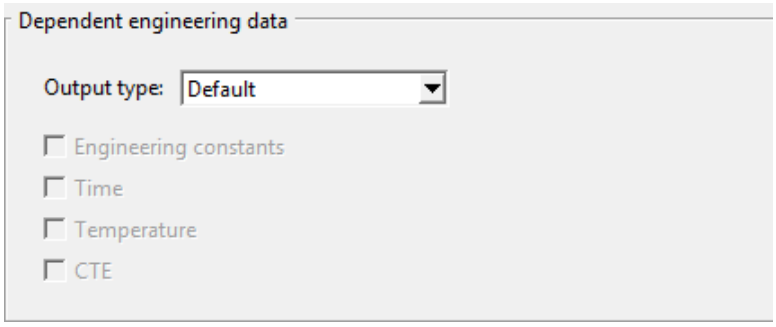


Figure 11-5 Output available for Dependency Elastic Moduli file.

The available output fields for this file are:

- Time: Time of the analysis
- Temperature: Temperature field
- AlphaGlob: Global expansion matrix
- Engineering constants: Engineering constants

Restrictions

- The global expansion matrix is only available for purely thermo-elastic composite, i.e., it is not available for thermo-elasto-plastic and thermo-elasto-viscoplastic matrix composite.
- Temperature field is only available for thermal analysis.

Output Available in the .dsn File(s)

This file contains the macroscopic stress amplitude and critical number of cycles allowing the user to plot the macroscopic S-N curve of the composite material. In case of multilayer RVE, per-ply `.dsn` files are also exported; they contain the macroscopic critical number of cycle and the failure indicator value computed over the corresponding layer. Three types of output are available:

- None: the Fatigue Results (`*.dsn`) file will not be created.
- Default: the default output fields will be printed.
- Custom: the user can choose which fields to output.

Restrictions

- These fields are only available for a fatigue analysis.

The available output fields for this file are:

- Sa: Stress amplitude (for pseudo-grain fatigue only)
- SaL: Longitudinal stress amplitude (for damage fatigue only)
- SaT: Transverse stress amplitude (for damage fatigue only)
- SaS: Shear stress amplitude (for damage fatigue only)
- Nc: Critical number of cycles
- Dc: Critical damage value
- FC: Failure indicator value (for pseudo-grain fatigue only)

Remark: The failure indicator outputs have different significations depending on the context:

- For single-layer RVE: The macroscopic DSN file contains {FC1, FC2... FCi} e.g. the average value of the failure indicator failure indicators {1, 2... i} over the RVE. At least one of these values shall be close or equal to 1, unless the result is unconverged (typically if Nc 1 or Nc - Ncendurance).
- For multi-layer RVE:
  - The macroscopic DSN file contains FC, e.g. the multilayer-failure definition of all fatigue failure indicators over the RVE (see subsection Failure for Multilayer RVE). This value shall be close or equal to 1, unless the result is unconverged.
  - For each layer, a per-layer DSN file is available; it contains {FC1, FC2... FCi} e.g. the average value of the failure indicators {1, 2... i} over the layer when macroscopic failure is reached. This output is mostly provided for investigation purpose, it enables to determine which layer(s) trigger (or participate the most to) macroscopic failure.

Output Available in the .log File

The user can follow the evolution of the residual during the loading of the three principal algorithms of Digimat-MF:

- Augmented-Lagrangian algorithm: AL / AL.res / AL.iter
- Free-stress algorithm: FS / FS.res / FS.iter
- Mori-Tanaka algorithm: MT / MT.res / MT.iter

Three types of output are available:

- None: Same behavior as Default
- Default: the default output fields will be printed. In this case, no information about algorithm will be written in the `*.log` file.
- Custom: the user can chose the fields to output.

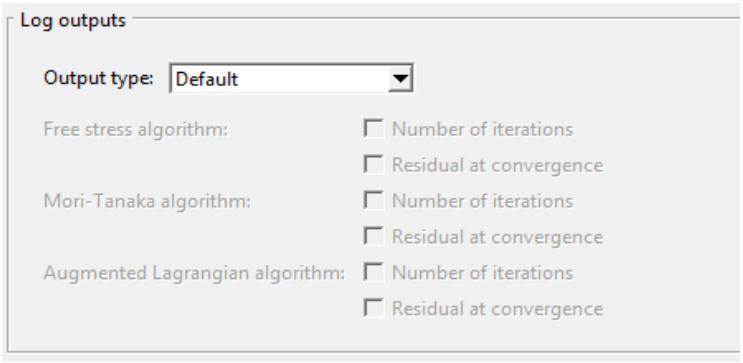


Figure 11-6 Output available for log file.

By default, all these information are not printed in the `*.log` file. If an orientation tensor is used at the inclusion level, information of Mori-Tanaka algorithm are the following:

- MT.res = MAX(MT.res) over each pseudo grains
- MT.iter = MAX(MT.iter) over each pseudo grains

Restrictions

- For a purely (thermo-)elastic composite, information of the convergence of both Mori-Tanaka algorithm and augmented-Lagrangian algorithm are invalid because:
  - augmented-Lagrangian method is only used to manage incompressibility of hyperelastic materials
  - no iterative homogenization scheme is used for this particular case; stress and strain in each phase are computed analytically, i.e., without any iterative algorithm.

- For the homogenization of composite involving thermal and electrical materials, none of aforementioned algorithms is available.
- If general 3D loading is used, free-stress algorithm is not available, i.e., no information about residual evolution and the number of iteration will be available
- For a 2-phase composite involving an orientation tensor, residual evolution of the MFH algorithm is not printed for each pseudo-grain; the maximum value of the residual over each pseudo-grain will be printed in the `*.log` file.
- For a N-phase composite, residual evolution of the MFH algorithm is not printed for each phase; the maximum value of the residual over each phase will be printed.

Numeric Precision in Output Files

By default, Digimat writes the numeric values using a scientific format with a relative precision of 5 digits; e.g., a value of 12.3456789 will be printed 1.2346e+001. However, in some cases, this precision is not sufficient to get a smooth aspect of the result curves. This is the case, for example, for the two following cases:

- Hyperelastic material (normally used with large deformations) with small strain amplitudes; in this case Digimat outputs the transformation tensor, $F = 1 + \varepsilon$, of which values are close to one;
- Creep and relaxation loading, for which the characteristic times related to the pre-loading and creep (resp. relaxation) stages are not in the same order of magnitude.

In such cases, the output precision can be increased (but not decreased) using any of the following settings:

- **DIGIMATMF_Output_Precision** field in the `DIGIMAT_Settings.ini` file (see section Use of DIGIMAT_Settings.ini File).

Use of DIGIMAT_Settings.ini File

This is a global setting which affects all the analyses created or run with Digimat-MF.

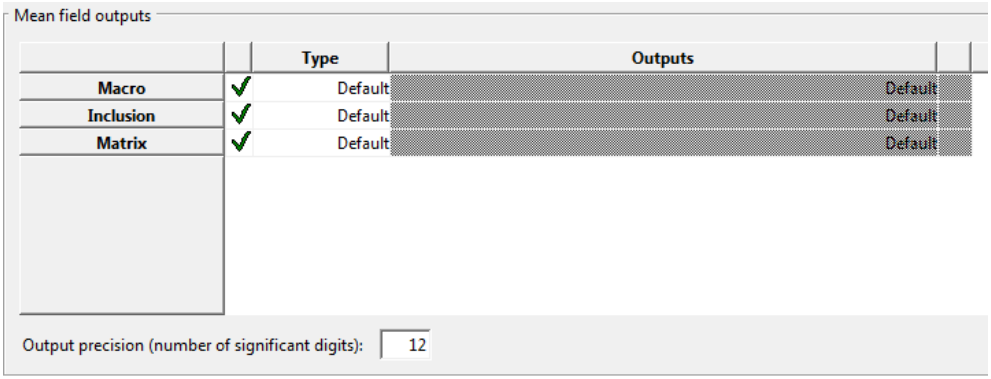


Figure 11-7 Setting the output precision in Digimat-MF.

- **In the Output tab**, the **Output precision** field of the Mean Field Outputs box (see Figure 11-7) defines the output precision for the current analysis only; if non-default, this setting overrides the **global** setting specified in the `DIGIMAT_Settings.ini` file.

Remark:

- The output precision setting affects all the Digimat output files (`*.mac`, `*.mtx`, `*.icl`, `*.ctg`, `*.dsn`, `*.dem`, `*.eng`) except the Digimat log file (`*.log`).
- This setting is only available in Digimat-MF.

Output Files and Variables

This section presents the various files and variables involved in analyses with Digimat-MF.

Output Files

Several ascii files are generated by Digimat-MF when running an analysis. These files contain various fields which are detailed here after. They can either be loaded in the graphical user interface or edited by the text editor of your choice.

All the files generated by Digimat-MF are named after the job and analysis names, according to the following template: `JobName_AnalysisName*.*`. Depending on the file name and extension, whether it is a small or finite strain analysis, various results are available for post-processing. See the Table 11-1 for the output variable sets.

- `JobName_Analysis.mat`: The Digimat-MF input deck file. It contains all the analysis information defined in the graphical user interface, i.e., the material parameters, the microstructure definition as well as the analysis parameters.

- `JobName_Analysis.log`: The Digimat-MF analysis log file. It echoes the Digimat-MF input deck and lists the analysis execution comments, e.g., analysis warning and error messages.
- `JobName_AnalysisName.mac`: The analysis results at the composite level, also referred to as the macroscopic level. The relevant fields to be listed in this file belong to Set 1 or Set 3.
- `JobName_AnalysisName_MatrixName.mtx`: The analysis results at the matrix phase level. These are average results throughout the phase. The relevant fields to be listed in this file belong to Set 2 or Set 3.
- `JobName_AnalysisName_InclusionName.icl`: The analysis results at the inclusion phase level. These are average results throughout the phase. The relevant fields to be listed in this file belong to Set 2 or Set 4.
- `JobName_AnalysisName_InclusionName_CoatingName.ctg`: The analysis results at the coating phase level. These are average results throughout the phase. The relevant fields to be listed in this file belong to Set 2.
- `JobName_AnalysisName_InclusionName_matrixInClusterName.ctg`: The analysis results at the matrix in the cluster. These are average results throughout the phase. The relevant fields to be listed in this file belong to Set 2.
- `JobName_AnalysisName_MatrixName_AllLayers.mtx`: The analysis results at the global matrix phase level. These are results averaged over all layers of the laminate material. The relevant fields to be listed in this file belong to Set 2.
- `JobName_AnalysisName.eng`: The macroscopic engineering constants of the composite material defined in the analysis. The relevant fields to be listed in this file belong to Set 5.
- `JobName_AnalysisName.dem`: The temperature-dependent macroscopic engineering constants. This file is only relevant for thermo-elastic linear analyses.
- `JobName_AnalysisName.dsn`: The macroscopic results relevant to plot the macroscopic S-N curve of the composite material. This file is only relevant for fatigue analyses. The relevant fields to be listed in this file belong to Set 6.

Remark: For a laminate material (multilayer analysis), the average per phase results are also available for each layer of the laminate.

Remark: For a rubber matrix composite reinforced with elastic inclusions having a non-fixed orientation inside the RVE (tensor-specified or random orientation), theta_a, phi_a, theta_b, phi_b, theta_c, phi_c are replaced by a_{11}, a_{22}, a_{33}, a_{12}, a_{23} and a_{13}, the components of the orientation tensor.

Table 11-1 Output variable sets for Digimat-MF analyses

Variable set	Available output fields
Set 1 - Small strain analysis	time, e_{11}, e_{22}, e_{33}, $2*e_{12}$, $2*e_{13}$, $2*e_{23}$, s_{11}, s_{22}, s_{33}, s_{12}, s_{13}, s_{23}, s_{eq}, temp
Set 2 - Small strain analysis	time, e_{11}, e_{22}, e_{33}, $2*e_{12}$, $2*e_{13}$, $2*e_{23}$, s_{11}, s_{22}, s_{33}, s_{12}, s_{13}, s_{23}, s_{eq}, temp, p, ep_{11}, ep_{22}, ep_{33}, $2*ep_{12}$, $2*ep_{13}$, $2*ep_{23}$, X_{11}, X_{22}, X_{33}, X_{12}, X_{13}, X_{23}
Set 3 - Finite strain analysis	time, F_{11}, F_{21}, F_{31}, F_{12}, F_{22}, F_{32}, F_{13}, F_{23}, F_{33}, Theta, s_{11}, s_{22} s_{33}, s_{12}, s_{23}, s_{13}, s_{eq}, Sn_{11}, Sn_{21}, Sn_{31}, Sn_{12}, Sn_{22}, Sn_{32}, Sn_{13}, Sn_{23}, Sn_{33}, NE_{11}, NE_{22}, NE_{33}, $2*NE_{12}$, $2*NE_{23}$, $2*NE_{13}$, E_{11}, E_{22}, E_{33}, $2*E_{12}$, $2*E_{23}$, $2*E_{13}$, I_1, I_2, I_3, I_{max}, ENER, temp
Set 4 - Finite strain analysis	time, F_{11}, F_{21}, F_{31}, F_{12}, F_{22}, F_{32}, F_{13}, F_{23}, F_{33}, Theta, s_{11}, s_{22}, s_{33}, s_{12}, s_{23}, s_{13}, s_{eq}, Sn_{11}, Sn_{21}, Sn_{31}, Sn_{12}, Sn_{22}, Sn_{32}, Sn_{13}, Sn_{23}, Sn_{33}, NE_{11}, NE_{22}, NE_{33}, $2*NE_{12}$, $2*NE_{23}$, $2*NE_{13}$, E_{11}, E_{22}, E_{33}, $2*E_{12}$, $2*E_{23}$, $2*E_{13}$, I_1, I_2, I_3, I_{max}, ENER, a, b, c, $theta_a$, phi_a, $theta_b$, phi_b, $theta_c$, phi_c, temp
Set 5 - Engineering constants	Stiffness, compliance & CTE matrices, Young's moduli, Poisson's ratio
Set 6 - Fatigue analysis	SaL, SaT, SaS, Nc, Dc

Output Variables

Here below are listed the definition of all the variables used in `.mac`, `.mtx` and `.icl` files:

- time: analysis time.
- e_{11}, e_{22}, e_{33}, $2*e_{12}$, $2*e_{23}$, $2*e_{13}$: components of the strain tensor.
- s_{11}, s_{22}, s_{33}, s_{12}, s_{23}, s_{13}: components of the Cauchy stress tensor.
- s_{eq}: von Mises equivalent stress.
- std_s: standard deviation on the matrix von Mises equivalent stress (only available for second order homogenization, in the matrix output file).
- s_{eq2}: second order von Mises equivalent stress (only available for second order homogenization, in the matrix output file).
- dam_D: damage parameter for Lemaître-Chaboche elasto-plastic model.
- P: accumulated plastic strain.
- ep_{11}, ep_{22}, ep_{33}, $2*ep_{12}$, $2*ep_{13}$, $2*ep_{23}$: components of the plastic strain tensor.
- X_{11}, X_{22}, X_{33}, X_{12}, X_{13}, X_{23}: components of the back stress tensor.

- temp: temperature.
- $Tgrad_1$, $Tgrad_2$, $Tgrad_3$: temperature gradient (only for thermal analyses).
- $Hflux_1$, $Hflux_2$, $Hflux_3$: heat flux (only for thermal analyses).
- $Vgrad_1$, $Vgrad_2$, $Vgrad_3$: voltage gradient (only for electrical analyses).
- J_1, J_2, J_3: current densities (only for electrical analyses).
- F_{11}, F_{21}, F_{31}, F_{12}, F_{22}, F_{32}, F_{13}, F_{23}, F_{33}: components of the deformation gradient.
- Theta: average volume change for incompressible materials. Note that for thermo-hyperelastic materials. Theta represents the average volume change associated with the elastic part of the deformation gradient.
- Sn_{11}, Sn_{21}, Sn_{31}, Sn_{12}, Sn_{22}, Sn_{32}, Sn_{13}, Sn_{23}, Sn_{33}: components of the nominal strain tensor.
- NE_{11}, NE_{22}, NE_{33}, $2*NE_{12}$, $2*NE_{23}$, $2*NE_{13}$: components of the nominal strain tensor.
- E_{11}, E22, E_{33}, $2*E_{12}$, $2*E_{23}$, $2*E_{13}$: components of the Green-Lagrange strain tensor.
- a, b, c: updated principal dimension of the deformed inclusion.
- $theta_a$, phi_a: spherical angles, in degrees, defining the orientation with respect to the RVE's coordinate system of the principal direction of the deformed inclusion corresponding to the principal dimension a.
- $theta_b$, phi_b: spherical angles, in degrees, defining the orientation with respect to the RVE's coordinate system of the principal direction of the deformed inclusion corresponding to the principal dimension b.
- $theta_c$, phi_c: spherical angles, in degrees, defining the orientation with respect to the RVE's coordinate system of the principal direction of the deformed inclusion corresponding to the principal dimension c.
- I_1, I_2, I_3: invariants of the Cauchy-Green strain tensor.
- IMAX: maximum of I_1, I_2, I_3.
- ENER: strain energy of an hyperelastic material.
- a_{11}, a_{22}, a_{33}, a_{12}, a_{23}, a_{13}: updated components of the orientation tensor.
- $f1_A$, $f1_B$, $f2_A$, $f2_B$, etc.: outputs for regular (non FPGF) failure indicators. $f1_A$ is the first output (A) of the first indicator (f1) and $f1_B$ is the second output (B) of the first indicator (f1). There is a maximum of five outputs, from A to E. There is no upper-limit on the number of failure indicators.
- D_{11}, D_{22}, D_{33}, D_{12}, D_{23}, D_{13}: damage variables for the progressive failure mechanism.
- PGA, PGB, PGC: output variables for the FPGF indicator. PGA is the weighted fraction of failed pseudo-grains, PGB is the relative number of failed pseudo-grains, and PGC is the average value of the (thresholded) failure indicator.

- FI_AVG and FI_THK: additional failure outputs for multilayer failure (only at macroscopic level, and when multilayer failure controls are meaningful). FI_AVG is the average value of the failure indicator, and FI_THK is the thickness fraction of failed layers.

The definition of all the variables used in `.eng` files are listed here:

- E_{11}, E_{22}, E_{33}: macroscopic Young's moduli.
- nu_{12}, nu_{21}, nu_{13}, nu_{31}, nu_{23}, nu_{32}: macroscopic Poisson's ratios.
- G_{12},G_{13},G_{23}: macroscopic shear moduli.
- K: macroscopic bulk modulus.
- a_{Axial}: macroscopic axial coefficient of thermal expansion (11).
- a_{OutPl}: macroscopic out-of-plane coefficient of thermal expansion (33).
- a_{InPl}: macroscopic in-plane coefficient of thermal expansion (22).

The definition of all the variables used in `.dsn` files are listed here:

- SaL, SaT & SaS: longitudinal (along direction 1 of the RVE), transverse (along direction 2 of the RVE) & shear stress amplitude.
- N_c: critical number of cycles.
- D_c: critical damage value.

Finite Strain Tensors

When working with hyperelastic materials in Digimat-MF, the finite strain formulation of continuum mechanics is employed. Here below are defined the finite strain tensors and the associated variables:

- Nominal stress tensor:

$$Sn = JF^{-1} \cdot \sigma \tag{11-3}$$

- Nominal strain tensor:

$$NE = V - I \tag{11-4}$$

- Green-Lagrange strain tensor:

$$E = \frac{1}{2}(F^T \cdot F - 1)$$

- Piola-Kirchhoff stress tensor:

$$P = Sn^T \tag{11-5}$$

- Deformation gradient tensor:

$$\mathrm{F} = \frac{\partial x}{\partial \mathrm{X}} \tag{11-6}$$

- Left stretch tensor, stemmed from the polar decomposition of the deformation gradient tensor:

$$\mathrm{F} = \mathrm{V} \cdot \mathrm{R} \tag{11-7}$$

- The Jacobian determinant of the motion or determinant of the deformation gradient tensor:

$$\mathrm{J} = \det \mathrm{F} \tag{11-8}$$

- The second order identity tensor:

$$\mathrm{I} = \mathrm{diag}(1, 1, 1) \tag{11-9}$$

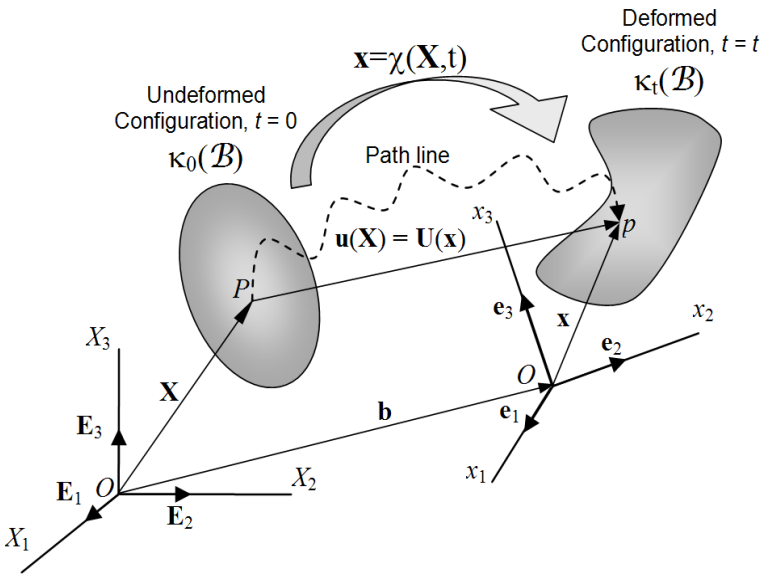


Figure 11-8 Transformation of a continuum.

Database Files

A set of files can be saved and loaded from and in the graphical user interface. These ascii files contain various information according to their importance in the Digimat material definition tree:

- Analysis.daf: Digimat analysis file: it contains all the parameters relative to a Digimat-MF analysis (material parameters, microstructure definition, analysis parameters, etc).
- Material.dmf: Digimat material file: it contains the parameters relative to the definition of a particular material.

- Phase.dpf: Digimat phase file: it contains all the parameters relative to the definition of a particular phase.
- FailureIndocator.dfi: Digimat failure indicator file: it contains all the parameters relative to the definition of a failure indicator - but not its assignment parameters: level (Composite/Phase) and mechanism (Standard/FPGF).

Plotting Tools

This section presents the actions that can be carried out from the plot item of the Digimat tree, where the outputs of a Digimat analysis can be plotted. See Output Files and Variables for more detail about which curves can be plotted. Figure 11-9 presents the graphical user interface (GUI) of Digimat, in particular the plot area.

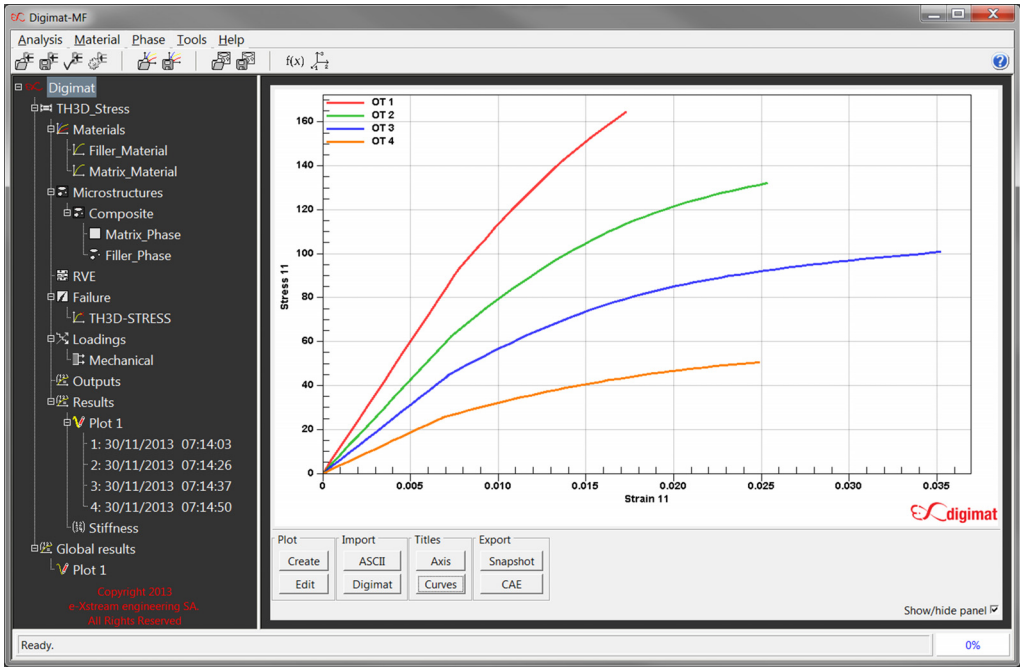


Figure 11-9 Graphical user interface of Digimat - Plot area.

Loading of Analysis Results

Before plotting any curves in the plot area, the results to be plotted need to be loaded in the GUI. By rightclicking on the **Plot *n*** item in the Digimat tree, where n is the plot number, the results named after the current analysis and job name can be loaded via the Load results option, provided they are available in the current working directory.

Note that, if several analyses are defined in the GUI, the job name is common for all the analyses defined in the Digimat tree, contrary to the analysis name which is used to differentiate them. The current analysis is the one that was last selected.

Plotting of Results

Once the results have been loaded in the GUI, the different output variables are available for plotting, one versus another. To access the plotting tools, the Plot n tree item should be selected. These are displayed at the bottom of the **viewport**, see Figure 11-10. They allow the user to perform several operations which are described hereafter. Some of them are not fully illustrated, as carrying them out is straightforward.

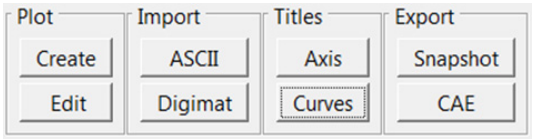


Figure 11-10 Plotting tools.

Plot - Create

This button opens a window containing the list of the results loaded in the GUI, see Figure 11-11. The user should then select which data, X Data, should plotted against which other data, Y Data. The interface window should be exited before carrying out any further manipulation in the plot area. This can be done via the **Dismiss** button.

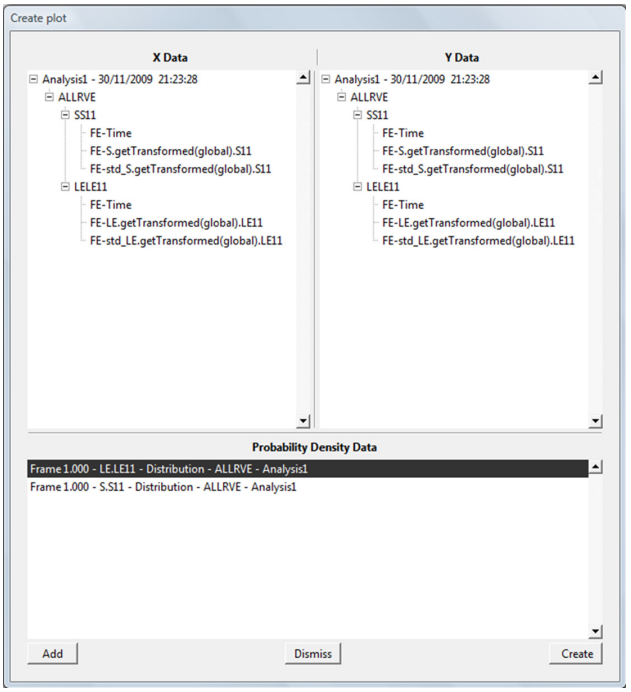


Figure 11-11 Create plot interface window.

Plot - Edit

This interface window, quite similar to the Create plot one, allows the removal and the replacement of the currently displayed curve in the plot area.

Import - ASCII

Tabular data stored in ascii files can be imported in the GUI for plotting in the plot area, provided the columns are separated either via a blank space or a tabulation.

Import - Digimat

This option allows the loading of a Digimat-MF result file, whether it is results at the composite level or at the phase level.

Titles - Axis

Clicking the button opens an interface window enabling the modification of the legend entries of each axis.

Titles - Curves

Clicking the button opens an interface window enabling the modification of the legend entries of each curve.

Export - Snapshot

Clicking the button creates a snapshot of the current window that is stored in memory and can easily be pasted into presentations or other documents.

Export - CAE

This option allows to export one stress strain curve plotted in the plot window to an elastoplastic material card to be used in a CAE tool. The following CAE tools are supported:

- Abaqus (`*.inp`)
- ANSYS (`*.cdb`)
- LS-DYNA (`*.k`)
- Marc (`*.dat`)
- PAM-CRASH
- RADIOSS Block.

In order to use this capability, there can only be one curve plotted in the plot window. The Young's modulus and yield stress are computed from the plotted curve. The Young's modulus is computed using the first two points of the curve. The yield stress is determined by finding the first point with a non linearity higher than 0.2 percent. All the points after the yield stress are exported as a piecewise linear isotropic hardening law.

A dialog box will prompt for the values of the Poisson's ratio and density, since it is not possible to deduce those parameters from a single stress - strain curve.

Plotting Failure Envelopes

Digimat-MF enables to compute and plot failure envelopes, which concept is explained in subsection General Concept. This can be performed from the **Failure envelope** tab, which appears under the **Results** item of the model tree (see Figure 11-12).

This capability is available for thermo-mechanical analyses including failure criteria, with a few limitations listed in subsection Failure Envelope.

You can also refer to the Frequently Asked Questions.

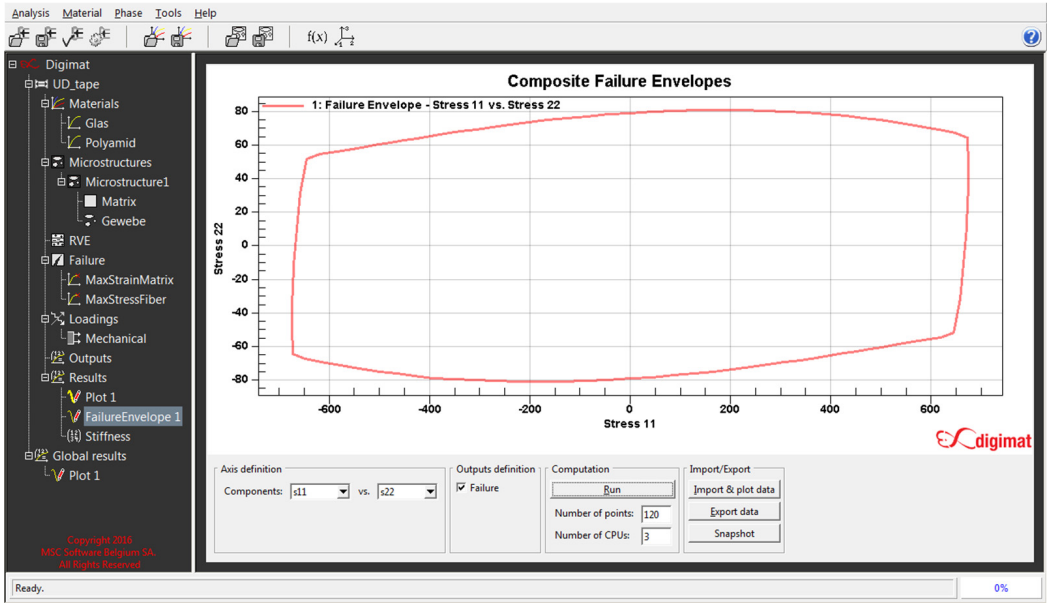


Figure 11-12 General aspect of the Failure envelope tab in Digimat-MF GUI.

General Concept

A 2D failure envelope is created, for a material with a constant microstructure, by computing the strengths for biaxial monotonic loadings, with various ratio between the two loading components. Digimat-MF enables to compute either stress-based or strain-based failure envelopes:

- For strain-based envelopes, the two quantities plotted are the maximal strains, and the loading is designed such that all the other strain components are null.
- For stress-based envelopes, the two quantities plotted are the stress at failure, and the loading is designed such that all the other stress components are null.

Remark: This means that stress-based and strain-based envelopes are not equivalent in terms of loading.

For example, a **S11 vs. S22** envelope assumes a plane stress state, whereas **E11 vs. E22** envelope assumes a plane strain state.

The choice of stress-based or strain-based type depends on the targeted usage, but in general:

- Stress-based envelopes provide easier comparison with experimental results, which generally use plane stress loading condition.

- Strain-based envelopes are easier to interpret for strain-based failure indicators with interaction between components (like Tsai-Hill class), or when the material behavior yields a strong stress saturation.

Failure envelopes are usually closed, e.g., failure is supposed to happen for every possible loading definition. This a physical consistency condition. Nevertheless, open failure surfaces are supported in Digimat-MF, for investigation purpose.

The concept of failure envelope can also be extended to other failure-related events, for example damage initiation or stress drop, which are meaningful for ductile failure. Such **extended** definitions are available in Digimat-MF and detailed in subsection Available Output Types.

Usage in Digimat-MF GUI

This section presents how to compute, display and plot failure envelopes in Digimat-MF.

Starting from a correct Digimat-MF analysis definition, a new **Failure envelope** item can be added by rightclicking on the **Results** item and selecting **Add failure envelope area**. Moreover, when loading an analysis file which meets the required conditions, a failure envelope plot area is automatically created.

The Controls ribbon at the bottom of the tab (see Figure 11-13) provide buttons and controls to define, compute and plot failure envelopes. The content of this ribbon is detailed hereafter.

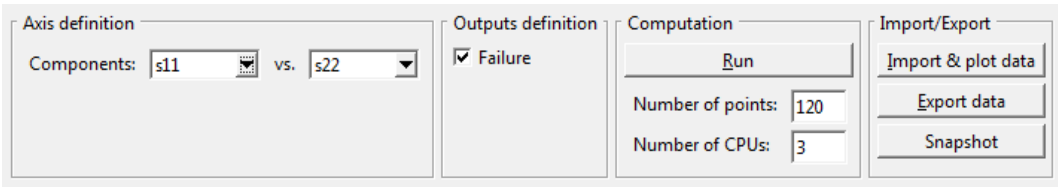


Figure 11-13 Failure envelope tab controls ribbon.

Axis Definition

The **Axis definition** groupbox provides controls to specify the loading and output components for the failure envelope (see Figure 11-14). The loading parameters defined here will be used for the computation of the failure envelope(s), and will override the loading definition given in the **Loading** tab.

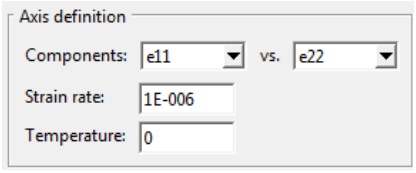


Figure 11-14 Failure envelope controls for loading definition.

In terms of mechanical loading direction, six options are available:

- S11 vs. S22
- S11 vs. S12
- S22 vs. S12
- E11 vs. E22
- E11 vs. 2×E12
- E22 vs. 2×E12

Each pair indicates the loading direction, but also the failure envelope type (stress-based or strain-based), as explained previously.

When strain-rate dependencies are defined over the mechanical properties or the failure strengths, an additional **strain rate** option appears. This option enables to define the loading strain rate to be applied, for both strain-based and stress-based envelopes. It enables to highlight the effect of strain-rate dependencies on failure behavior.

Remark: In this case, the loading strain rate is based on the tensorial norm of the strain, which corresponds to the definition being used for strain rate dependencies (see subsection Strain Rate-dependent Mechanical Parameters). This definition differs from the one proposed in the Loading tab, which is based on a single strain component.

When the analysis type is thermo-mechanical, the additional **Temperature** field that appears defines the temperature of the isothermal mechanical loadings. This enables to highlight the effect of temperature dependencies on failure behavior.

Available Output Types

The **output** groupbox gives the possibility to select one or several failure envelope types (see Figure 11-15):

- When a standard or FPGF failure criterion is assigned, only the **Failure** output is available:
  - When the RVE is monolayer (classical), the failure envelope corresponds to the instant when the maximum failure indicator reaches the critical value.
  - When the RVE is mutlilayer, the failure envelope corresponds to the instant when the multilayer failure criterion is reached, following the definition set in the Failure assignment tab (see subsection Failure for Multilayer RVE).
- When a progressive failure criterion is assigned, two outputs are available:
  - The **damage initiation** envelope corresponds to the instant when any of the progressive failure damage variables become non-null.
  - The **stress drop** envelope corresponds to the instant when of the stress components reach a maximum, for the two directions considered in output.

Remark: These two envelopes coincide when the damage law is instantaneous, and may differ when the damage is very progressive.

Figure 11-15 Failure envelope outputs for standard/FPGF failure (left), and for progressive failure (right).

Failure Envelope Computation

The **computation** groupbox provides controls and buttons for computing the failure envelope(s). The user may define the number of points plotted on the envelope, and also the number of processors to use for computation. Depending on the number of points, processors, and material definition, computation time may vary between 5 seconds and several minutes.

The evaluation is numerical: multiple Digimat-MF analyses are run, and the corresponding strain/stress components at failure are extracted. The loading type is automatically determined, as well as the optimal loading range which enables to reach failure for all scenario.

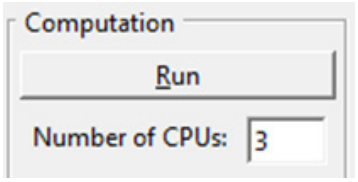


Figure 11-16 Failure envelope computation controls.

Remark: In failure is not reached for a given scenario (or damage initiation and stress drop in case of progressive failure), Digimat will re-run it with a larger loading range; this is performed a limited number of times, in case the failure definition is intentionally meant to yield an open failure surface. However, such open failure surface may appear non-smooth, due to time incrementation-related error. This generally indicates an improperly-defined failure indicator, but may also happen with progressive failure if a **no-damage law** is used, or when the stress level never drops.

The evaluation operation will write several files to the current working directory:

- `JobName_AnalysisName.mat`: The Digimat-MF input deck file, which contains additional keywords related to the failure envelope computation.
- `JobName_AnalysisName.log`: The failure envelope computation log file. It echoes the execution comments, e.g., analysis warning and error messages.
- `JobName_AnalysisName_EnvelopeType.dfe`: The computation results for a given failure envelope type.

The file contains two data columns, corresponding to the two stress/strain components selected in the **Axis definition** groupbox.

These file are not deleted after computation, so they can be reused afterwards.

After a successful computation, the failure envelope results are automatically imported and added to the current plot area.

Plot and Export Results

The **Import/Export** groupbox provides buttons to import, plot and export results, and to snapshot the graphic window.

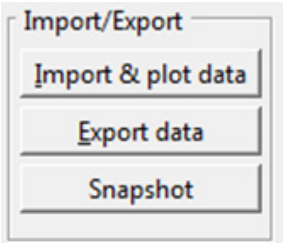


Figure 11-17 Failure envelope import/export buttons.

The **Import & plot** button enables to import and plot two types of results:

- Digimat Failure Envelope files (`*.dfe`), typically generated by previous computations.
- Digimat Experimental Files (`*.def`) and ASCII files (`*.txt`), which should contain two data columns; each row can be considered as a data point with a (x,y) pair, defining the first and second stress/strain components to be plotted. More info on the formatting of such files can be found in subsection Experimental Data Files.

After selecting a file through the dialog box that pops up, the data is automatically plotted as a new curve in the graphic area. By default, experimental data curves are plotted with symbols instead of lines, but this can easily be changed in the Customization dialog (right-click on the graphic area).

The graphic area can be cleared by right-clicking on the **Failure envelope** item in the model tree. It is currently not possible to remove curves individually, nor to change the plot legends and axis names.

Remark: The format of Digimat Failure Envelope files is compatible with other Digimat-MF results files, so it is possible to import and plot those files in a regular Plot tab, which provides more controls.

The **Export data** button enables to export the currently plotted curves to Digimat Failure Envelope files. Clicking on the button pops up a dialogbox which enables to select one or several curves to export, and the filename to use for each one (see Figure 11-18).

Remark: The output file is always formatted as a Digimat Failure Envelope file (`*.dfe`), independently of the original data source.

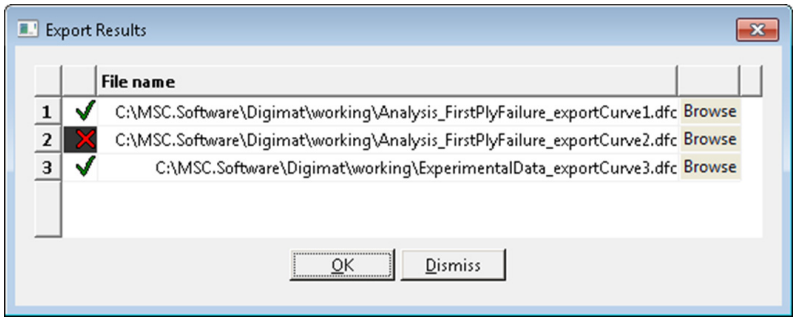


Figure 11-18 Failure envelope results export dialog box.

Frequently Asked Questions

- The computation of a failure envelope takes too much time, how can I reduce it?

  Computation time usually vary between 5 seconds and several minutes, and depends mostly of the following parameters (by order of significance):

  - the number of processors used for computation;
  - the number of points requested;
  - the number of layers, if the RVE is multilayer;
  - the "behavior complexity" in general: Digimat-MF computations are faster for an elastic behavior than for a viscoelastic-viscoplastic behavior.
  - the time required to check-in a Digimat-MF license, which may be significant for floating licenses if the port is incorrectly specified.

- Why are the points not regularly spaced on my failure envelope?

  This can happen if the failure indicator is strongly anisotropic, or has strong tension-compression differentiation. The loading scenarios are generated following a regular rectangular-shaped distribution, in the stress or strain space (depending on the plotted quantity). This choice can be more or less optimal depending of the failure indicator expression and the plotted quantity (see Figure 11-19).

- Can I plot failure envelopes with a user-defined failure criterion?

  Yes, you can, as long as the failure indicator outputs are not negative. Furthermore, the failure envelope will have a smoother aspect if these outputs evolve continuously w.r.t. stress or strain.

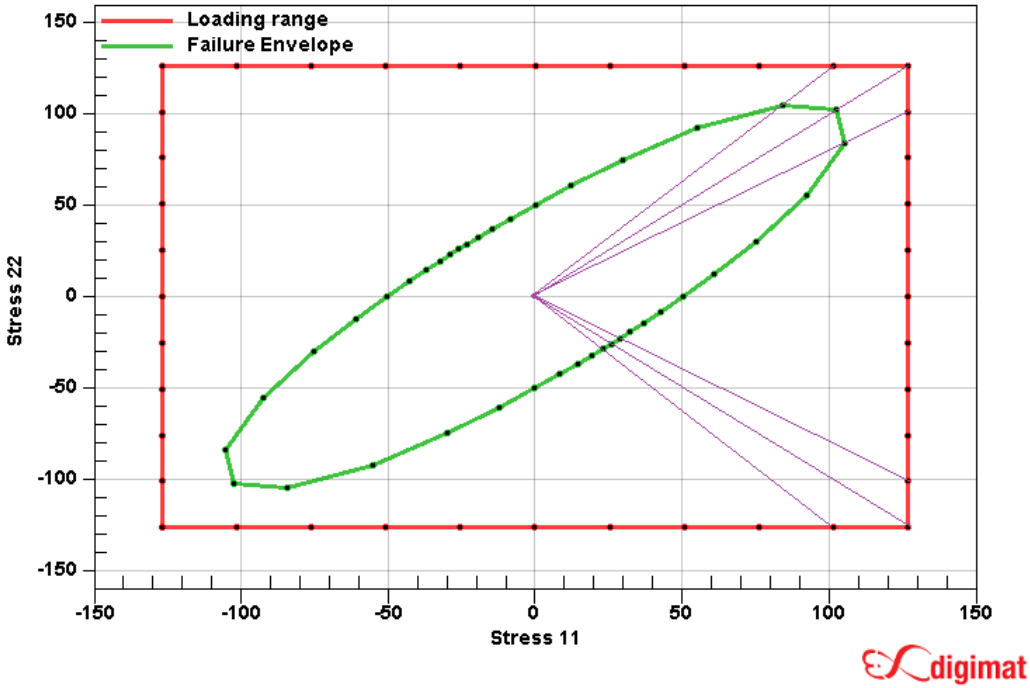


Figure 11-19 Failure envelope with non-regular points spacing.

Figure 11-19 shows Failure envelope with non-regular points spacing. The failure criterion is Tsai-Hill strain based, but the failure envelope is stress-based.

Plotting Carpet Plots

Digimat-MF enables to compute and display carpet plots, which concept is explained in CMH-17 and in subsection General Concept. This can be performed from the **Carpet Plot** tab, which appears under the **Results** item of the model tree (see Figure 11-20).

This capability is available for thermo-mechanical material using either a 2-phase UD microstructure or a basic woven, with a few limitations listed in subsection Carpet Plot.

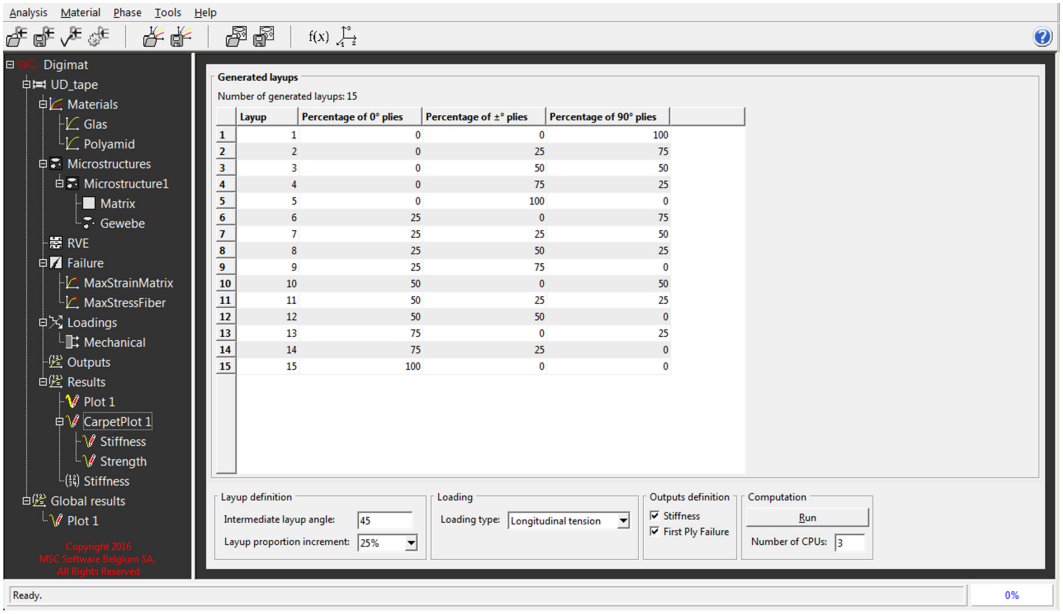


Figure 11-20 General aspect of the Carpet plot tab in the Digimat-MF GUI.

General Concept

A carpet plot is created by computing the uniaxial strength for various stackings composed of different fiber orientation, and plotting this value against the respective proportions of these orientations (see Figure 11-21). Such plots are valuable for the selection of an appropriate laminate stacking sequence, during the criticalstrength and stiffness design steps.

Carpet plots can be computed for different failure-related events (first-ply failure, last-ply failure, damage initiation), or for other mechanical properties like the apparent stiffness.

They can also be plotted for intermediate angles different from $\pm 45°$, or for different ply microstructures (like woven), as long as all plies have the same microstructural properties.

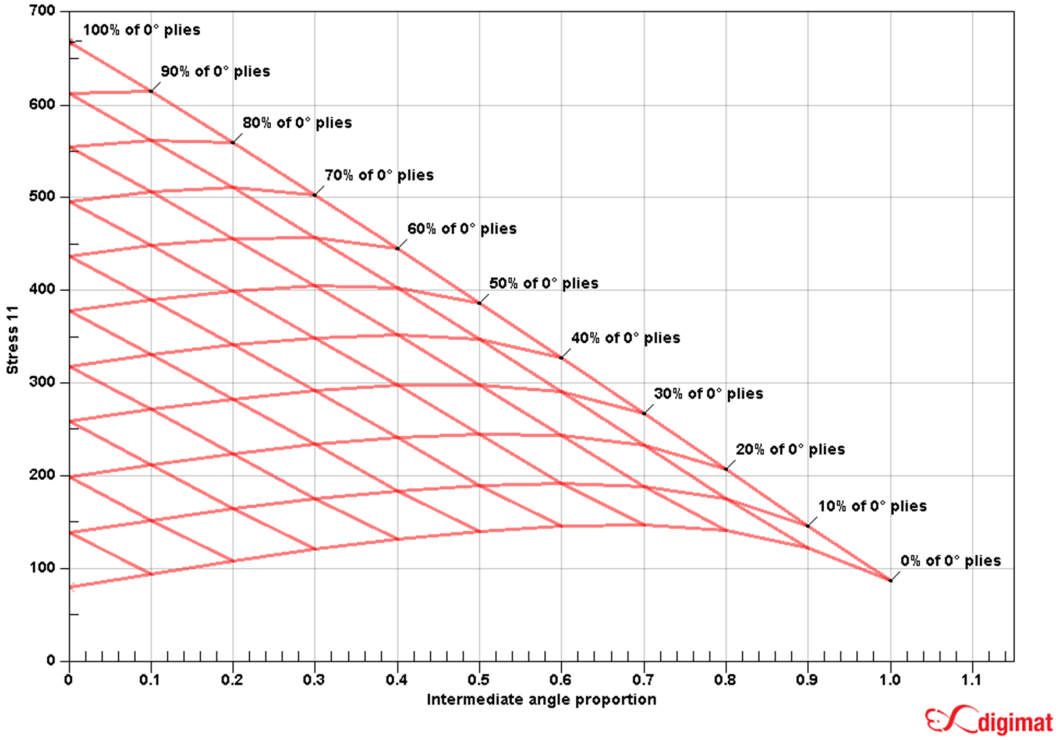


Figure 11-21 Tensile strength of $[0_i/\pm 45_j/90_k]_S$ family of glass/polyamid laminates.

Usage in Digimat-MF GUI

This section presents how to compute and display carpet plots in Digimat-MF GUI.

Digimat-MF enables to compute first-ply failure and apparent stiffness carpets plots, for symmetric and balanced laminates based on unidirectionnal plies (e.g. with continuous fibers).

Starting from a correct Digimat-MF analysis definition, a new **Carpet plot** item can be added by right-clicking on the **Results** item and selecting **Add carpet plot area**. Moreover, when loading an analysis file which meets the required conditions, a carpet plot area is automatically created.

The Controls ribbon at the bottom of the tab (see Figure 11-22) provide buttons and controls to define, compute and display carpet plots. The content of this ribbon is detailed hereafter.

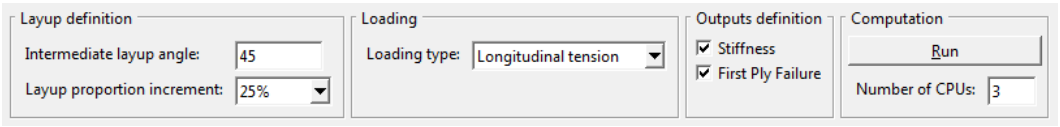


Figure 11-22 Carpet plot tab controls ribbon.

Figure 11-22 shows carpet plot tab controls ribbon. Different controls may appear, depending on the material and failure definition.

Layup Definition

Digimat-MF computes carpet plots based on the $[0_i/\pm\theta_j/90_k]$s family of UD laminates. The **Layup definition** groupbox enables to define the intermediate angle and the **layup proportion increment** which defines the number of layups to be generated. The main tab provides a preview of the orientation proportions of the layups that will be generated.

Remark:

- The + and - ply proportions are counted together. For example, a $[0/\pm 45_2/90]$s has 25% 0° ply proportion, 50% ±45° ply proportion, and 25% 90° ply proportion.
- When the microstructure is a unidirectional ply, its orientation is overriden to $\{\theta = 90, \varphi = 0\}$, before application of layer additional rotations; on the other hand, when the microstructure is woven, its orientation is not overriden before application of layer additional rotations;

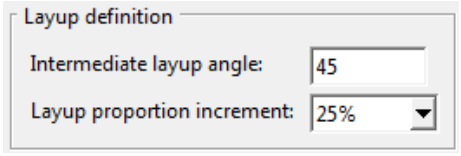


Figure 11-23 Carpet plot layup definition controls.

Loadings

The **Loading** groupbox provides controls to specify the loading applied to carpet plot computation (see Figure 11-24). The loading parameters defined here will be used for the computation of the carpet plot(s), and will override the loading definition given in the **Loading** tab.

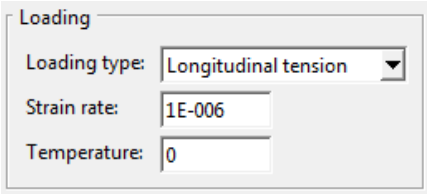


Figure 11-24 Carpet plot loading controls.

The mechanical loading is strain-based and uniaxial. It can be either:

- Longitudinal tension, e.g. uniaxial tension with $\{\theta_{load}=90,\ \varphi_{load}=0\}$
- Longitudinal compression
- Transverse tension, e.g. uniaxial tension with $\{\theta_{load}=90,\ \varphi_{load}=90\}$
- Transverse compression
- In-plane shear

When strain-rate dependencies are defined over the mechanical properties or the failure strengths, an additional **strain rate** option appears. This option enables to define the loading strain rate to be applied, for both strain-based and stress-based envelopes. I enables to highlight the effect of strain-rate dependencies on failure behavior.

Remark: In this case, the loading strain rate is based on the tensorial norm of the strain, which corresponds to the definition being used for strain rate dependencies (see Chapter 3: Setting Dependencies). This definition differs from the one proposed in the Loading tab, which is based on a single strain component.

When the analysis type is thermo-mechanical, the additional **Temperature** field that appears defines the temperature of the isothermal mechanical loadings. This enables to highlight the effect of temperature dependencies on the stiffness and the failure behavior.

Carpet Plot Types

The **output** groupbox gives the possibility to select among the two following carpet plot types (see Figure 11-25):

- The apparent stiffness in the loading direction.

  This "apparent" stiffness is simply defined as the ratio of the stress component and the strain component in the loading direction, at the first time increment (e.g. in the elastic domain).
- The first-ply failure stress in the loading direction.

  This output is only available when one (or several) standard failure indicators are assigned at the phase level. This is currently the only supported failure configuration, see subsection Carpet Plot for more detailed explanations.

Remark: It is not possible to compute and plot carpet plots based on the strain at failure. Although this is conceptually valid, the corresponding graphics are barely interpretable.

As the quantities corresponding to these outputs are not comparable (not expressed in the same units), they are plotted in two separate windows (see Figure 11-5).

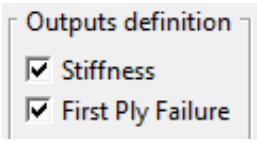


Figure 11-25 Carpet plot output types.

Carpet Plots Computation

The **Run** button in the **Computation** groupbox launches the evaluation of the carpet plot. The user may also specify the number of processors to use for computation.

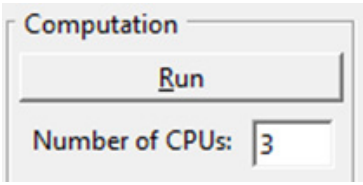


Figure 11-26 Carpet plot computation controls.

The evaluation is numerical: multiple Digimat-MF analyses are run (with various laminate stackings), and the corresponding apparent stiffness and/or strengths are extracted. The loading type is automatically determined, as well as the optimal loading range which enables to reach failure for all scenarii (using the same strategy as for failure envelopes, see Failure Envelope Computation).

The evaluation operation will write several files to the current working directory:

- `JobName_AnalysisName.mat`: The Digimat-MF input deck file, which contains additional keywords related to the carpet plot computation.
- `JobName_AnalysisName.log`: The carpet plot computation log file. It echoes the execution comments, e.g., analysis warning and error messages.
- `JobName_AnalysisName_CarpetPlotType.dfc`: The computation results for a given carpet plot type.

  The Digimat Failure Carpet files (`*.dfc`) contain three data columns standing for, respectively:

  - the stacking formula
  - the proportions of the $0°/\pm/90°$ ply orientations

- the strength or the apparent stiffness modulus

These file are not deleted after computation, so they can be reused afterwards.

After a successful computation, the carpet plot results are automatically imported and added to their respective plot sub-items.

Display, Import and Export Results

The carpet plots are plotted in two separate graphic areas, that can be displayed by clicking on the **Strength** and **Stiffness** sub-items in the model tree (see Figure 11-27). These graphic tabs provide a slightly different controls ribbon, containing the basic layups definition controls (for information only) and some additional buttons to import, plot and export results, and to snapshot the graphic window.

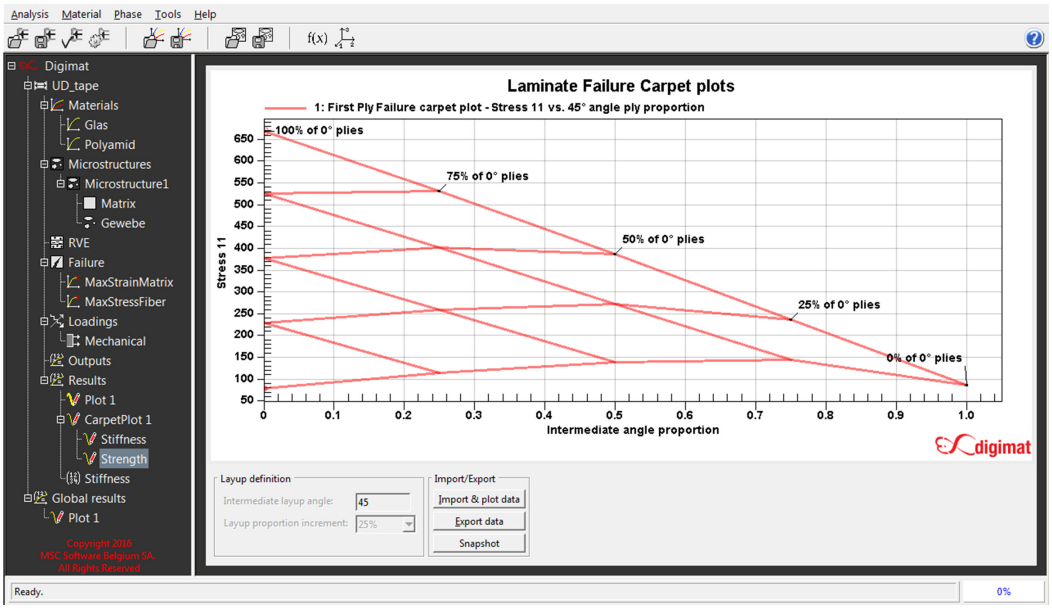


Figure 11-27 Carpet plot graphic plot areas.

The **Import & plot** button enables to import and plot two types of results:

- Digimat Failure Carpet files (`*.dfc`), typically generated by previous computations.

  The carpet plot type (stiffness or strength) is automatically determined from the file name suffix (_Stiffness or _FirstPlyFailure), and the new curve is plotted in the appropriate graphic area.

- Digimat Experimental Files (`*.def`) and ASCII files (`*.txt`), which should contain two spaced-separated data columns. The first column can be either:

  - the proportions of ± plies, or

- a series of stacking formulas formatted like in `*.dfc` files, e.g. for example:
  [0_2/45_1/-45_1/90_2]_s, [0_2,90_4,0_2], or [0]_8s

The second column should provide the stiffness or the strength.

The new curve is always plotted in the current graphic area.

By default, experimental data curves are plotted with symbols instead of lines, but this can easily be changed in the Customization dialog (right-click on the graphic area).

The graphic area(s) can be cleared by right-clicking on the **Carpet plot** item or sub-item in the model tree. It is currently not possible to remove curves individually, nor to change the plot legends and axis names.

The **Export data** button enables to export the currently plotted curves to Digimat Failure Carpet files. Clicking on the button pops up a dialogbox which enables to select one or several curves to export, and the filename to use for each one (see Figure 11-28).

Remark: The output file is always formatted as a Digimat Failure Carpet file (`*.dfc`), independently of the original data source.

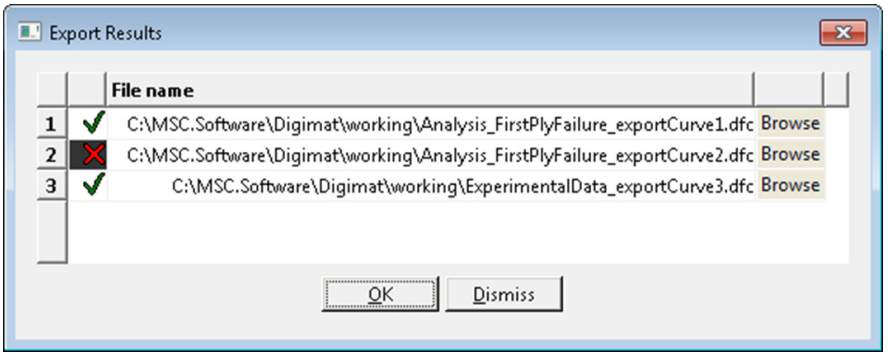


Figure 11-28 Carpet plot results export dialog box.

Plotting Forming Limit Diagrams

Digimat-MF enables to compute and plot forming limit diagrams (FLD), which concept is explained hereafter.

Such computation can be performed from the **Forming Limit Diagram** tab, appearing under the **Results** item of the model tree (see Figure 11-29).

This capability is only available for mechanical analyses involving metal microstructure with plasticity or crystal plasticity without failure.

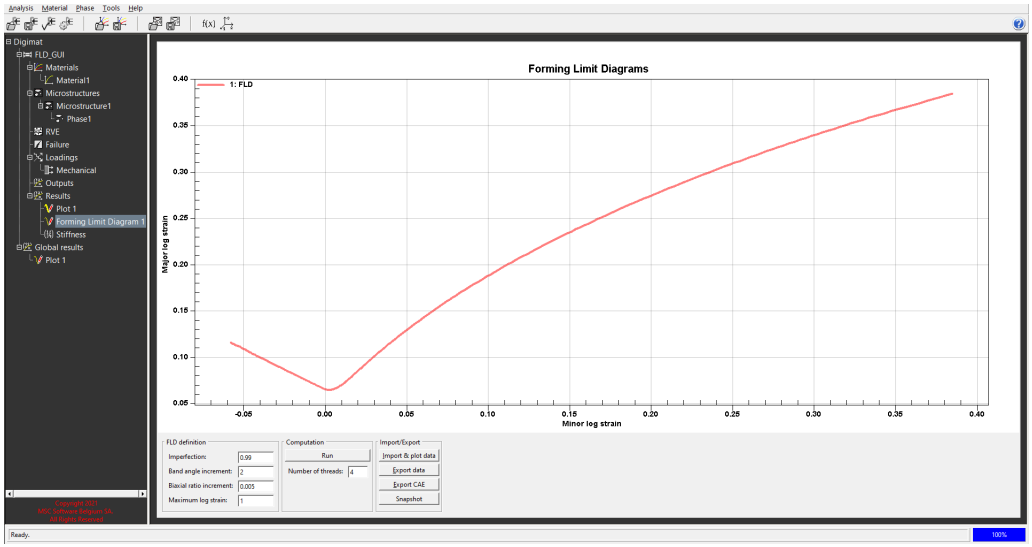


Figure 11-29 Forming Limit Diagram tab in Digimat-MF GUI.

General Concept

The prediction of plastic localization is of prime interest for industrial metal forming processes in order to evaluate metal sheet formability (i.e. the capability of sheet metal to deform plastically to a given shape without plastic instabilities nor failure). The localization of plasticity is an instability which leads to a transition from uniform to non-uniform mode of deformation while the loading remains uniform. Two types of instabilities are distinguished: diffuse necking and localized necking. Upon loading, diffuse necking initiates when the geometric softening due to local area reduction of the sheet overcomes strain hardening. The neck develops progressively and considerable extension is still possible after the onset of diffuse necking. Upon additional deformation, the necking region further localizes until a sharp localized band starts to form.

Various theoretical necking models have been proposed in the literature to predict the formability of metallic sheets characterized by the so-called forming limit diagram (FLD), representing the major and the minor strains corresponding to necking for any monotonic proportional strain paths. In practice, FLD is a very useful tool which enables for instance, to determine

the range of safety for deep-drawing, the critical zones where necking or fracture are most likely to occur, or the favourable working conditions (blank holding, lubrication etc.).

In Digimat-MF, the localized necking analysis is based on the Marciniak and Kuczynski (1967) model that postulates the existence of a material imperfection such as a groove or a narrow band with a slightly diminished thickness across the width of the metal sheet as depicted in Figure 11-30 In its modified form proposed by Hutchinson and Neale (1978), the band is oriented at an angle ϕ with respect to the tensile direction x_1. The quantities inside and outside the band are denoted by

the superscripts *b* and *a*, respectively. The imperfection factor *f* is defined as the ratio of the initial thickness inside and outside the band $f = \frac{h_0^b}{h_0^a}$.

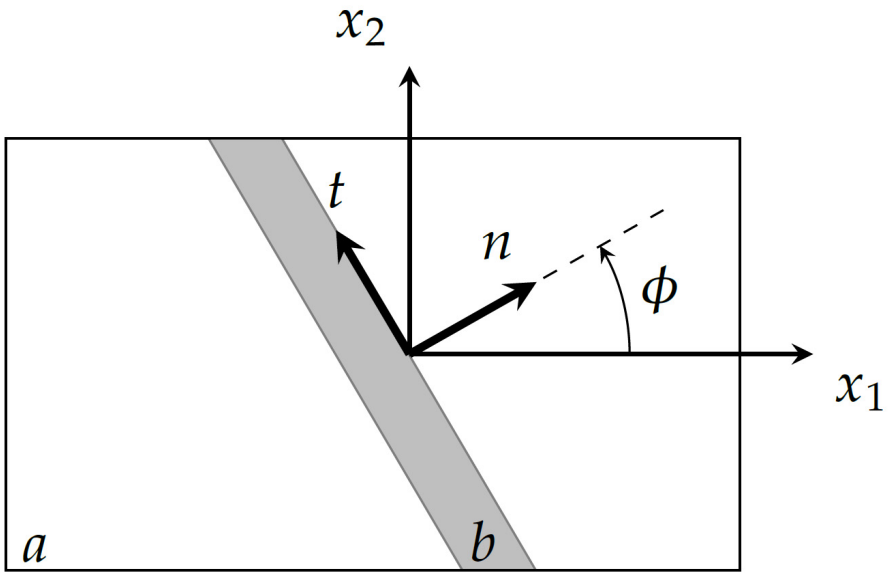


Figure 11-30 Sheet with an initial thickness imperfection band (b) oriented at an angle ϕ with respect to the major loading direction.

The band orientation differs from its initial state, ϕ_0 and varies with the deformation outside the band which is assumed homogeneous.

$$tan(\phi) = \frac{F_{11}^a}{F_{22}^a} tan(\phi_0) \tag{11-10}$$

where the deformation gradient *F* is computed from the velocity gradient $L = F\dot{F}^{-1}$.

Following Wu et al. (1997), the velocity gradient tensor components inside the band are expressed by:

$$L_{ij}^b = L_{ij}^a + \dot{c}_i n_j \quad for\ i, j = 1, 2 \tag{11-11}$$

$$L_{33}^b = L_{33}^a + \dot{c}_3 \tag{11-12}$$

where n is the band normal direction. The heterogeneity of the velocity gradient is thus introduced using the variables $\dot{c}$. These variables are determined by enforcing that the band out-of-plane stress is nil, and by balancing forces across the planar interface between the sheet and the band in the current configuration:

$$n_i \sigma_{ij}^b h^b = n_i \sigma_{ij}^a h^a \tag{11-13}$$

where the thickness of each zone evolves according the out-of-plane deformation: $h^a = h_0^a F_{33}^a$ and $h^b = h_0^b F_{33}^b$.

The onset of the localized necking is assumed to occur when the thickness strain rate is much larger inside the band than outside. The following necking criterion is used:

$$\left| D_{33}^b / D_{33}^a \right| > 10 \tag{11-14}$$

This point of instability is characterized by a given value of the sheet elongation F_{11}^a which is expected to depend on the initial inclination of the band ϕ_0. All initial band orientation ϕ_0 between 0 and 90 degrees are considered and the localization strain (i.e. one point of the FLD) is defined as the earliest occurrence of necking.

The localization model is solved numerically using an implicit time integration scheme.

Usage in Digimat-MF GUI

This section presents how to compute, display and plot FLDs in Digimat-MF.

Starting from a correct Digimat-MF analysis definition, a new **Forming Limit Diagram** item can be added by right-clicking on the **Results** item and selecting **Add FLD plot area**.

Moreover, when loading an analysis file which meets the required conditions, a FLD area is automatically created.

The Controls ribbon at the bottom of the tab (see Figure 11-31) provide buttons and controls to define, compute and plot FLDs. The content of this ribbon is detailed hereafter.

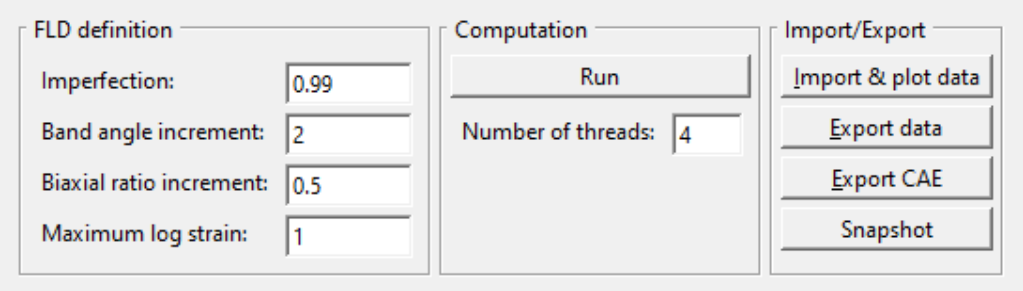


Figure 11-31 FLD tab controls ribbon. Different controls may appear, depending on the material definition.

FLD Definition

The **FLD definition** groupbox provides controls to specify the loading and localization model parameters for the FLD.

The loading parameters defined here will be used for the computation of the FLD, and will override the loading definition given in the Loading tab. The user may enter the following parameters:

- **Imperfection**: specify the imperfection factor used in the localization model.
- **Band angle increment**: specify the increment of band angles (in degrees), used to find the earliest necking in the localization analysis.
- **Biaxial ratio increment**: specify the increment of biaxial strain ratio, used to evaluate different points of the FLD.
- **Maximum log strain**: specify the maximum logarithmic peak strain, after which the localization model is no more evaluated.

FLD Computation

The **computation** groupbox provides controls and buttons for computing the FLD.

The user may define the number of processors to use for computation.

Depending on the band angle increment, biaxial strain ratio increment, processors, and material definition, computation time may vary between 5 seconds and several hours.

The evaluation is numerical: multiple Digimat-MF analyses are run, and the corresponding minor and major strain/stress components at necking are extracted.

The evaluation operation will write several files to the current working directory:

- `JobName\_AnalysisName.xml` – The Digimat-MF input deck file,

which contains additional keywords related to the FLD computation.

- `JobName\_AnalysisName.log` – The FLD computation log file.

It echoes the execution comments, e.g., analysis warning and error messages.

- `JobName\_AnalysisName.dfl` - The computation results for a given FLD.

The file contains two data columns, corresponding to the minor and major logarithmic strains.

These file are not deleted after computation, so they can be reused afterwards.

After a successful computation, the FLD results are automatically imported and added to the current plot area.

Plot and Export Results

The **Import/Export** groupbox provides buttons to import, plot and export results, and to snapshot the graphic window.

The **Import & plot** button enables to import and plot two types of results:

- Digimat Forming limit diagram files (`*.dfl`), typically generated by previous computations.
- ASCII files (`*.txt`), which should contain two data columns;

each row can be considered as a data point with a (x,y) pair, defining the minor and major strain to be plotted.

After selecting a file through the dialog box that pops up, the data is automatically plotted as a new curve in the graphic area.

By default, experimental data curves are plotted with symbols instead of lines, but this can easily be changed in the Customization dialog (right-click on the graphic area).

The graphic area can be cleared by right-clicking on the **FLD** item in the model tree.

It is currently not possible to remove curves individually, nor to change the plot legends and axis names. The format of Digimat FLD files is still compatible with other Digimat-MF results files, so it is possible to import and plot those files in a regular Plot tab, which provides more controls.

The **Export data** button enables to export the currently plotted curves to Digimat FLD files.

Clicking on the button pops up a dialogbox which enables to select one or several curves to export, and the filename to use for each one. The output file is always formatted as a Digimat FLD file (`*.dfl`), independently of the original data source.

The **Export CAE** button enables to export the currently plotted curves to CAE input files. Currently supported CAE codes are:

- Marc
- FTI
- Abaqus

- LS-DYNA

Engineering Results

Besides providing time-based field results, Digimat-MF also computes macroscopic engineering results whenever possible. For mechanical analyses involving elastic materials, or materials having an elasto-plastic or an elasto-viscoplastic matrix with elastic inclusions, such results are available. It is also the case for thermal and electrical analyses.

Based on the tensorial results, Digimat-MF identifies the macroscopic material symmetries. In addition to the tensorial results expressed in the local and global axes, engineering constants are provided when symmetries are present. For short fibers and wovens, it is extracted from the stiffness in the local and global axes, respectively.

Tensorial Results, in Global and Local Axes

When performing analyses with Digimat-MF, tensorial results are computed. The considered tensors depend on the chosen analysis type. See Table 11-2 for the detail of the available results versus the selected analysis type.

Note that if some dependencies are defined in the analysis, Digimat outputs the tensorial data at the last computation increment. To observe the dependency, the appropriate variable can be plotted in the plot area.

Table 11-2 Tensorial output as a function of the analysis type

Analysis type	Tensorial output
Mechanical	Stiffness, Compliance
Thermo-mechanical	Stiffness, Compliance, CTE
Electrical	Electrical conductivity and resistivity
Thermal	Thermal conductivity and resistivity

By definition, tensors are mathematical entities independent of the basis used to represent them. Their representation is however directly dependent on the chosen basis. In Digimat, tensorial data can be expressed either in the global (RVE) or in the local (fibers) axes and represented by matrices.

The stiffness/compliance tensor being a fourth-order symmetric tensor, it is represented by 6-by-6 matrix, while the CTE/conductivity/resistivity tensor being a second order tensor, it is represented using a 3-by-3 matrix. The storage conventions for the stiffness/compliance tensors are the following ones:

- Strain storage convention:

$$\varepsilon = \{\varepsilon_{11}, \varepsilon_{22}, \varepsilon_{33}, 2\varepsilon_{12}, 2\varepsilon_{23}, 2\varepsilon_{13}\} \tag{11-15}$$

- Stress storage convention:

$$\sigma = \{\sigma_{11}, \sigma_{22}, \sigma_{33}, \sigma_{12}, \sigma_{23}, \sigma_{13}\} \tag{11-16}$$

Axes

Two different axis systems are available in Digimat-MF, the global and the local axis systems. The global axis system refers to the material point axis system, the axis system with respect to which the inclusion orientation is defined as well as the loading directions, while the local axis system is related to the orientation of the inclusion phase.

There are several definitions of the local axis system, depending on the definition of the inclusion orientations:

- Fixed orientation: the first local axis, the 1-axis, is oriented along the fiber direction, while the 2- and 3-axes are taken in a place perpendicular to the fiber direction.
- Random 2D orientation: the fibers are randomly oriented in the local (1,2)-plane. Axes 1 and 2 are taken orthogonal in this plane, while the 3-axis is taken normal to it. The 1 and 2 local axes are taken parallel to the 1 and 2 global axes.
- Orientation tensor: the local axis system is defined by the eigenvectors of the orientation tensors, the 1-axis being parallel to the eigenvector corresponding to the largest eigenvalue, the 3-axis to the lowest one. In case the orientation tensor has two identical eigenvalues, i.e., the composite material is transversely isotropic, the transverse direction is given by the eigenvector corresponding to the multiplicity one eigenvalue.

Note:	If dependencies on the material properties are defined, the engineering results are given at the last time increment.

(Thermo-)mechanical Analysis

In the case of mechanical analyses, Digimat-MF computes the stiffness and compliance matrices both in the local and global axis systems. These results are available in the Global axes and Local axes tabs of the Stiffness item in the Digimat tree. In addition to these matrices, engineering constants are given in the **homonymous** tab. The engineering constants are one set of independent parameters necessary to define Hooke's elasticity tensor. They are defined in the section Linear (thermo-)elasticity relative to (thermo-)elastic materials.

Note that for orthotropic materials, the spherical angles defining the local axes of symmetries are given as well: θ_i and φ_i, $i = 1, 2, 3$. See section Axis Systems for the description of the spherical angles.

The matrix of the coefficients of thermal expansion is also computed in the case of thermomechanical analyses.

When materials with strain rate or temperature dependent properties are used, Digimat computes the stiffness and compliance matrices for different times or temperatures. In that case, it is possible to choose which set of matrices should be displayed: the matrices for the initial or final time or temperature.

In the specific case of harmonic loadings, additional information is displayed. The real part as well as the imaginary part of the usual results are computed: stiffness and compliance matrices both in local and global axis systems, and engineering constants. These results are available at each frequency of the defined loading.

Thermal Analysis

For thermal analyses, Digimat-MF computes both the specific heat of the composite material and its thermal conductivity and resistivity tensors. The tensors are available in the **Global axes** and **Local axes** tabs while the **Engineering constants** tab contains the homogenized specific heat and conductivities if symmetries have been identified.

Electrical Analysis

For electrical analyses, Digimat-MF computes the electrical conductivity and resistivity tensors for the composite material. The tensors are available in the **Global axes** and **Local axes** tabs while the **Engineering constants** tab contains the homogenized conductivities if symmetries have been identified.

12 Specific Features

- Second Aspect Ratio

Second Aspect Ratio

This development aims at modeling flat fibers which requires two aspect ratio. A second aspect ratio is added in the Shape parameter description for both inclusion and void phase. The second aspect ratio is the ratio between the width and the height of the inclusion.

The material model defined with a second aspect ratio can be used in Hybrid and in structural applications. It can also be calibrated in Digimat-MX. The functionality has been internally validated with elastic, viscoelastic, thermoelastic, elastoplastic, thermoelastoplastic and elastoviscoplastic material behavior for static and crash application. It is known that the functionality does not behave properly with viscoelastic material model for creep application.

Regarding structural applications, the assumption is made that the fibers are flat in the x-y axis considering that the z axis is the one of the part thickness.

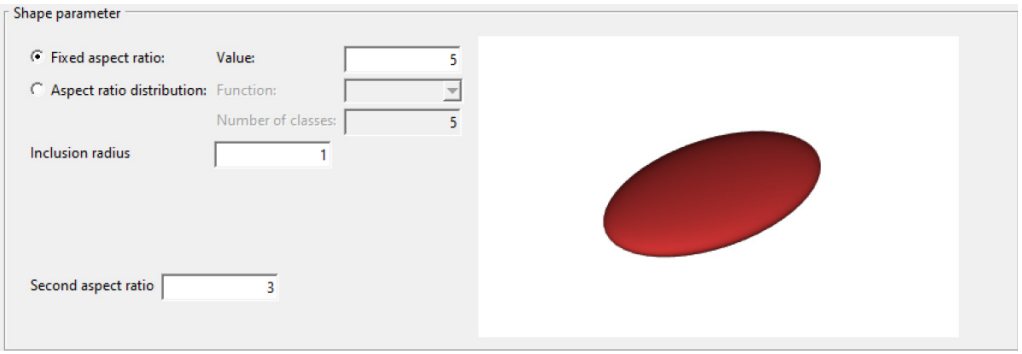


Figure 12-1 Illustration of an ellipsoid with two aspect ratio.

13 Guidelines

- Analysis Parameters & Convergence Issues
- Materials
- Failure
- Loadings
- Reverse Engineering of CTE

Analysis Parameters & Convergence Issues

When computing the response of a composite to a given loading, the **mean-field homogenization** field should always be checked on. If it is checked off, a Digimat warning should appear to inform the user of the eventual mistake the user could have done.

To get better computation accuracy it is recommended to increase the **number of angle increments** to 12. This recommendation should be respected if there are FPGF failure indicators defined in the analysis.

Strain Rate Dependency

For strain rate dependent computations, remind that **final time** actually triggers the strain rate of the analysis. When the user specifies some strain rate on the applied loading, Digimat automatically updates the final time such the strain rate is respected. The minimum and maximum time increments are also adapted in proportion to the correction applied on the final time.

When applying a **user-defined loading** (i.e., loading function), the user should manually adapt the final time and the time increments to account for the user-defined time history loading.

With strain rate dependent material models the **Use quasi-static loading** option (located on the **Loading -Mechanical** tab) does not verify after the computation that the loading is really quasi-static. This means, this option does not check that an additional decrease in strain rate doesn't affect anymore the stresses. Such a quasi-static strain rate depends on the material properties used in the strain rate dependent materials. It is therefore advised to set a low strain rate when defining the mechanical loading and to check that an additional decrease of the strain rate does not affect the stress level.

Convergence Issues

Convergence difficulties start to be seen when one of the following warning messages appear:

- `# DIGIMAT WARNING: Homogenization solution has been accepted (slow convergence)`
- `# DIGIMAT WARNING: Free boundary scheme solution has been accepted (slow convergence)`

When seeing those, it means that the **target tolerance** cannot be met under the permitted number or iterations respectively for the **homogenization** or the **loading scheme** (see section Homogenization Scheme Control & Loading Equilibrium Control). In such case, Digimat verifies if the schemes can at least meet the **acceptable tolerance**. Here is an example in which the acceptable tolerance can be respected but not the target one:

```
# Step = 1
# Increment = 23
# Current time = 0.361426
# Current time increment = 0.0266968
```

```
# Element number = 38749
# Integration point number = 3
# Section integration point number = 1
# Layer number = 1
# Current iteration number : 9
# Maximum number of iterations : 20
# Last residual : 1.07757e-04 > target tolerance = 1e-06 but < acceptable tolerance
= 1e-03
# Target tolerance : 1e-06
# Acceptable tolerance : 1e-03
# DIGIMAT WARNING : Homogenization solution has been accepted (slow convergence).
```

Sometimes the **last residual** becomes too high and the convergence issues can lead to trigger a time step reduction if it is an implicit run, or to terminate the run if it is an explicit run. Many possible reasons can explain that from which we can list too highly distorted elements or highly localized strain state where the stiffness gets almost null. Each case must be treated independently and it is impossible to diagnose all possible causes in a general way.

Materials

Isotropic Extraction Method

Which method to use?

Historically, the first method that has been proposed is the General method. This method is however not advised when working the J_2-plasticity model. It has been observed that General method might predict too stiff behaviors for composite, assuming stiff elastic fibers, and can lead to convergence issues in the Digimat-CAE computations. The **Spectral** and **Modified spectral methods** are to be preferred for J_2-plasticity matrices reinforced by stiff elastic fibers, e.g., a thermoplastic reinforced by glass fibers, though, if the matrix presents little hardening, the spectral method might still give too stiff predictions.

The modified spectral method is thus the most efficient method, though it involves four more parameters, which can be obtained via a fitting procedure of the stress/strain curves on the experimentally measured curves. These four additional degrees of freedom allow an accurate capture of the plastic behavior although their determination by fitting procedure may turn out complex and difficult.

The spectral method is an easier choice, more efficient than the general one and less complex to use than modified spectral, but can show its limits to model some plastic behavior such as for glass fiber reinforced thermoplastics.

Which parameters of the modified spectral should be adjusted?

The first parameter to modify to adjust the modified spectral method is the plastic strain multiplier, k_p. Experience shows that this parameter should not exceed a value of 10 in most cases. Higher values can lead to convergence issues in Digimat-CAE computations. Figure 13-1 illustrates the influence of the modified spectral method parameters on the composite response.

Note that the plastic strain multiplier, k_p, is the parameter that has the strongest influence on the composite response.

If adjusting the plastic strain multiplier does not allow an accurate capture of the plastic response, the plastic strain shift and the global shear multiplier can be adjusted. The plastic strain shift should stay reasonably small in comparison with the multiplied accumulated plastic strain $k_p p$, while the global shear multiplier should stay lower than the plastic strain multiplier. These advices are again based on experience and not on a theoretical proof.

Finally, it is advised to keep the plastic shear multiplier, k_t, equal to 1. Also note that different sets of parameters can lead to very close stress/strain responses. This problem of unicity, common to all inverse methods, may become an issue when adjusting these parameters.

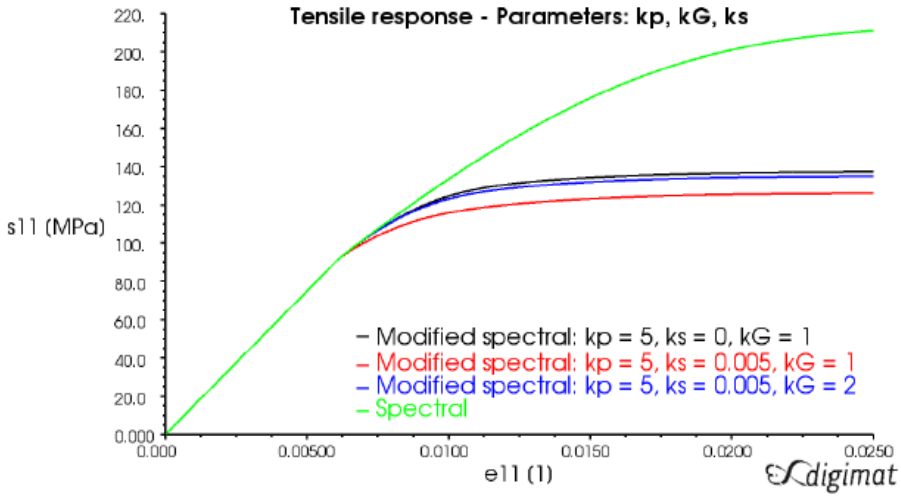


Figure 13-1 Influence of the modified spectral parameters on the composite tensile response.

Linear Thermo-elasticity

- The use of linear (thermo-)elasticity should be restricted to small strain, unless the material shows a linear behavior until failure.
- In general, the material parameters are temperature-dependent. If no temperature-dependence is defined for these, the temperature variation should be kept small.

Elastoplasticity

Which method to use for the isotropic extraction?

The isotropic extraction method that must be used by default is the spectral method. General method is usually not advised because it might predict a too stiff behavior in some cases, such as a J_2-elasto-plastic matrix with a little hardening reinforced with elastic stiff fibers.

It will be almost impossible to capture a plateau in the plastic region with this method, while a plateau can be easily captured with the spectral method. The modified spectral method is even more effective to capture the plateau than the original spectral but needs, at least, two additional parameters. This is the main disadvantage of the modified spectral method.

Exponential and linear hardening law vs exponential law

It is not always required to use an exponential and linear law to predict accurately the behavior of your material in the plastic region. Exponential law can be sufficient but can lead to convergence issues when some finite elements undergo plastic strain located far in the plateau region of your stress-strain curve.

In the plateau, the strain can increase indefinitely without variation of stress. This renders the convergence of the FE computation very difficult. By adding a linear term to the exponential law, an increase of stress is imposed for any increase of plastic strain. This will help the convergence of the FE computation.

It is therefore useful to use the exponential and linear hardening law even if the exponential law seems sufficient. Very small values for the linear term can be used (for instance 1 to 5 MPa for a matrix with a yield stress around 10MPa). The effect of the linear term on the accuracy of the predictions can be easily estimated by using Digimat-MF.

Elastoplasticity: Generalized Drucker-Prager

As only the general isotropic extraction method is available for this material model, some instability may occur during the homogenization procedure if the material tensile response presents a horizontal plateau. To minimize such problems, linear hardening may be introduced in the material law.

Note: This issue also arises with the classical Von Mises plasticity model.

Elastoplasticity: Damage Material

How to experimentally determine the damage model parameters?

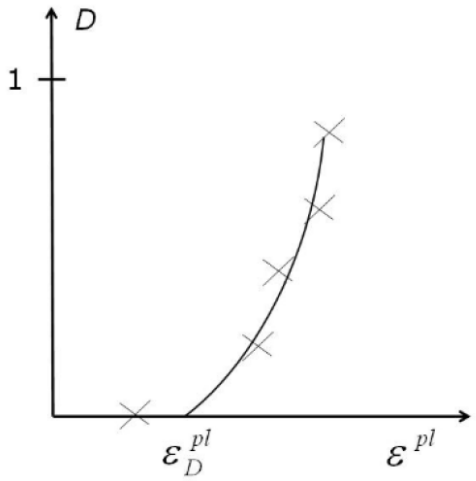


Figure 13-2 Experimental evolution of D vs. numerical prediction.

The curve shown in Figure 13-2 can be determined experimentally by loading and unloading the matrix material for increasing levels of plastic deformation. During each unloading step, the unloading slope (damaged Young modulus *E*) and plastic strain are recorded. The damage parameter *D* is obtained from

$$D = 1 - \frac{E_D}{E} \tag{13-1}$$

giving one point of the curve indicated by a cross. The three material parameters damage initiation threshold, damage rate factor and damage exponent *n*, are then determined such that the simulated curve (solid line in Figure 13-2) fits the experimentally determined curve (set of crosses) as closely as possible.

Why convergence issues can happen during damage in Digimat-CAE?

Damage can lead to strain softening of the material response. Such a behavior constitutes an instability of the material model, as the deformation increases indefinitely as long as a stress is applied on the material point. This instability translates mathematically into a negative tangent stiffness matrix which can lead to numerical convergence issues.

How to control the convexity of *D(p)* with the damage exponent?

The sign of the damage exponent *n* can be used to adjust the convexity of the $D(p)$ curve, as shown in Figure 13-3.

A positive value for the damage exponent *n* results in an exponential-like $D(p)$ curve (black), while a negative value for *n* results in a logarithm-like curve (red).

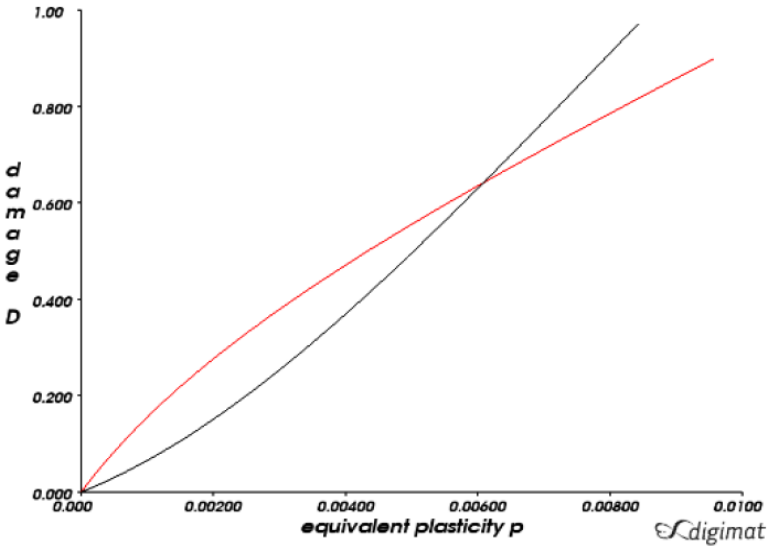


Figure 13-3 Effect of the damage exponent *n* on the convexity of *D(p)*.

Thermo-elastoplasticity

Guidelines are the same as for elasto-plasticity model (see section Elastoplasticity).

Elasto-viscoplasticity

- The discrete affine linearization method should be used on composites consisting of an elasto-viscoplastic matrix and an elastic inclusion phase.
- For creep loading, we advise to use the interaction law scheme to better describe the strain evolution.

Thermo-elastoviscoplasticity

Guidelines are the same as for elasto-viscoplasticity model (see section Elasto-viscoplasticity).

Viscoelasticity

- In the particular case of a composite made of a viscoelastic matrix reinforced with elastic inclusions surrounded with a coating phase, we advise to use the multi-level method as homogenization method.

- The main difficulty for viscoelastic materials consists in the determination of the parameters, such as the initial shear and bulk moduli G_0 and K_0 respectively, and the coefficients for the Prony series, i.e., the relaxation times and the Prony weights. These parameters are evaluated according to experimental data.

Stress/strain curves at different strain rate are not well adapted to define Prony series. Relaxation tests on a specimen are the most popular test to evaluate such parameters. It is naturally advised to use relaxation tests at different strain rates to calibrate a viscoelastic model. This experimental test is divided into two parts, as presented in Figure 13-4. First, a loading step and, second, a stabilization step during which the macroscopic strain is kept constant until all viscous effects have disappeared, i.e., until the stress vs. time curve tends to a horizontal asymptote.

Figure 13-5 presents the response of the RVE using Prony series containing 3 to 5 terms, the kept terms being the first ones of the example. As the relaxation times are chosen such that there is one by decade, it can be seen that the relaxation process takes longer in the 5-term case than in the 3-term one.

Accordingly, for an optimal viscoelastic procedure, the user will first use relaxation data on the material to identify the number of relaxation times or Prony series terms required to capture the step-like pattern of the stress vs. time curve (in log-log axes) and then adjust the Prony weights to have a good fit. Hu et al. (2006) provide more information as regards the identification of viscoelastic parameters.

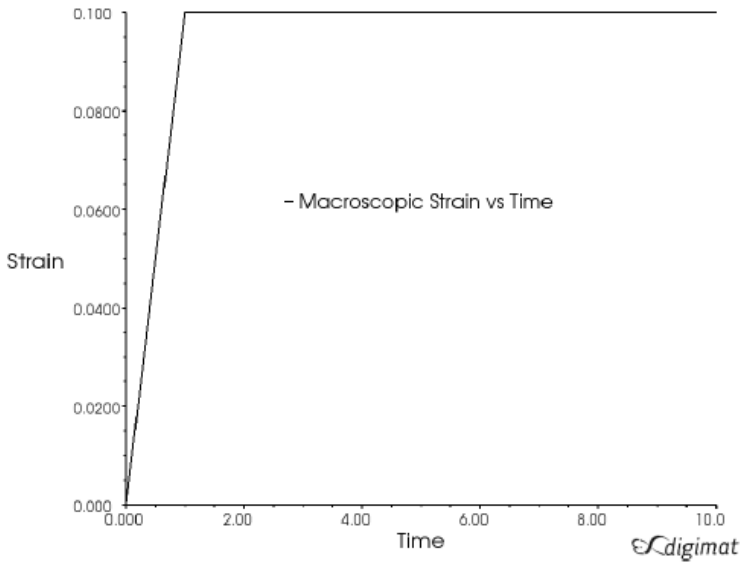


Figure 13-4 Macroscopic strain evolution as a function of time.

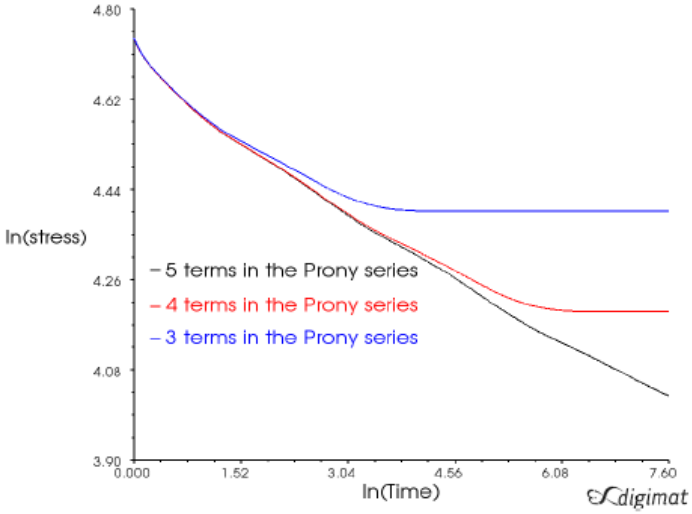


Figure 13-5 Stress-logarithm evolution as a function of the time logarithm for different number of time relaxation in the Prony series.

Viscoelasto-viscoplasticity

Yield Stress Determination

It has been demonstrated that elastic and viscoelastic deformations are totally reversible while plastic and viscoplastic deformations are permanent. This is a fundamental differentiation since viscoelastic-viscoplastic materials often show a nonlinear behavior in the elastic domain that makes the identification of the threshold between the elastic and the plastic zones, i.e., the yield stress, much harder to make.

In order to identify the yield stress, the best experimental approach is to make cyclic tests with a complete unload phase and sufficient time to let the material relax. The idea is, for every loading cycle, to slightly increase the peak stress applied. As long as the material entirely recovers after a certain relaxation time of the unloading phase, it means the deformation is reversible. In other words the material continues to deform in its viscoelastic regime. However, as soon as permanent deformation is observed after the unloading phase, which indicates the last load applied exceeded the yield stress. If the increase in the peak force increment is not too big, the peak stress reached for the last cycle can thus be interpreted as the yield stress of this material.

Thermo-hyperelasticity

For a two-phase composite where each phase is quasi-incompressible, it is advised to apply the augmented Lagrangian method only on the softer phase and not on both phases.

It is advised not to include geometric stiffness terms in a coupled finite element analysis, to improve convergence of the finite element. The exception is when elastic or rigid inclusions are used.

There are some recommendations on the use of the volume change parameter η. When (quasi-) incompressible hyperelastic materials are used in the homogenization scheme:

- If void inclusions are used, it is advised to compute η using the **incremental method** for the matrix phase (no material is defined as void).
- For thermal analyses, it is advised to compute η using the **implicit method** for each thermo-hyperelastic phase.
- For the other cases, the **explicit method** should be used to compute η; this is the default method.

Leonov EGP

- The set of parameters given in the example section comes from the literature on PC and is a good starting point to perform analyses using the Leonov-EGP model.
- By default, the Leonov-EGP model uses the neo-Hookean non viscous stress to compute the hardening stress.
- The homogenization procedure is performed using a theory similar to the one used for hyperelastic materials. Also, if the Leonov-EGP model is mixed with an hyperelastic material or another Leonov- EGP model, the user can follow the microstructure evolution during the loading through the evolution of the inclusion aspect ratio and orientation.
- During a coupled finite element analysis, the user is advised to set the option **Compute geometrical stiffness** to off, to improve the convergence of the analysis.

Failure

User-defined Failure Indicator

User's Examples

As a starting point for user implementation, some implementation examples are provided in the *Digimat examples manual*:

- Example of a max-strain/max-stress component failure indicator (in C++)
- Example of a Strain-Invariant failure indicator (in C++ and Fortran)
- Counter-example of a badly implemented failure indicator (in C++)

- Example of a Hashin 3D failure indicator used with Progressive Failure (in C++ and Fortran)

It is recommended to try running these examples, and re-compiling the libraries from the given source files, before trying to implement your own failure indicator.

Compiling Source Files as Dynamic Libraries

In order to package your code in a Dynamic Library file (`.dll` or `.so` file), you need to install a compiler for the programming language you're using.

- For Linux implementation, use of GCC compiler package is recommended
- For windows implementation, use of Visual Studio 2017 is recommended. It is available at: http://www.visualstudio.com/fr-fr/products/visual-studio-express-vs.
- For Python implementations, you can find a free compiler at: https://www.python.org/download/releases/2.6/.

The compiler options will be different depending on your operating system. After installation, do not forget to check that the folder containing the compiler binaries is present in your PATH system variable.

In order to inspect your DLL after compilation, you could also make use of some dependency inspection tools, like the Dependency Walker tool on Windows, or the `ldd` command on Linux.

C++ Implementation

Assuming you performed a C++ implementation in the files {myFailureCriterion.cpp, myFailureCriterion.h}:

- **Under GCC** Linux (x64):

```
g++ -m64 -shared -fPIC -o myFailureCriterion.so
myFailureCriterion.cpp
```

- **Using Visual Studio**:

  For Windows (x64): Create your project in visual studio and change its configuration type to **Dynamic Library (`*.dll`)**. You may have to disable incremental linking for the compilation to work: Rightclick on the project -> Properties -> Configuration properties -> Linker -> General -> Enable incremental linking = No. Launch Visual Studio (2017 or higher), then create and use an x64 configuration in the Configuration manager.

Fortran90 Implementation

Assuming you performed a F90 implementation in the file {myFailureCriterion.f90}:

- **Under GCC** Linux (x64):

```
gfortran -m64 -shared -fno - underscoring -fPIC
myFailureCriterion.f90 -o myFailureCriterion.so
```

- **Using the Intel Compiler** (Windows):

```
ifort /dll myFailureCriterion.f90 / out myFailureCriterion.dll
```

Frequently Asked Questions

What does the "-m64" compilation argument stand for?

This argument tells the compiler to build a 64-bits DLL, e.g. which can be used on 64-bits (a.k.a x64) operating system. Since Digimat is now supported on 64-bits operating systems only, make sure to use this compilation option.

What happens if the path to the DLL file is incorrect?

The Digimat GUI sends a warning message if the library file cannot be found on the computer, but the analysis can still run successfully (e.g. a material/interface file can be created). On the other hand, is the batch analysis fails, you should investigate the log file for corresponding error messages.

How should I prepare my analysis to make it usable on multiple machines? / How can I run a Digimat-CAE analysis (with user-defined failure indicator) on a remote machine?

When preparing a Digimat-CAE analysis to be run on a remote machine, we recommend you to use the **environment variable** option to make your analysis file configuration-independent. Alternatively, you can generate your interface files with an explicit path, then manually edit the `library_path` keyword the *.mat file (or even remove it to activate the **environment variable** option).

Moreover, the Digimat kernel smartly manages the library file extension, so you can specify the `library_path` with a `*.so` or a `*.dll` extension, or even no extension at all: this will be automatically corrected at run time, depending of the operating system.

My Digimat analysis fails with an error message like "Digimat could not load the library pointed by...". What should I do?

In case such an error message is sent during a Digimat analysis, please perform the following checks:

- If you use the `DIGIMAT2USUB_SHARED_LIBS` variable, check that is was correctly defined and that it points to a correct path. Reboot you computer if necessary. If it is correct, try to use the **explicit path** option instead: the **environment variable** option sometimes generates such kind of issues, depending of your user configuration and administrator rights.
- If you use a custom path, check that this path is not empty, and that is correctly taken into account by Digimat (try a dummy path and see if the error message changes in the log).
- Check that the library path (as echoed in the log file) points to an existing library file.
- Check that your implementation is correct, and that it fits the prototype provided in the template.

- Check that the compiler-specific redistribuables are provided in the PATH environment **variable.** Each compiler provides a bunch of resources (typically DLLs) that are needed for using the compiled binaries. Make sure that the user of your DLL implementation has installed these redistribuables on his machine, and that they are provided in his PATH. The only exception to this rule is for C++ implementations created with Visual Studio, since the related redistribuables are provided with the Digimat installation anyway.
- If your DLL implementation requires additional ressources (often referred as "dependencies"), please make sure that they are also added the `PATH` system variable. Dependencies can be easily investigated with tools like DependencyWalker on Windows, or the "`ldd`" command on Linux.
- If all these checks did not solve your problem, please contact digimat.support@hexagon.com for further help.

Are there any limitations over the failure functions that I shall implement?

Even though the user-defined failure function can (in principle) have any form, it is recommended to ensure that the failure indicator outputs remain greater than or equal to zero, and that at least one of these outputs can reach 1 for every possible stress-strain input. This means that the failure surface remains closed in every situation. This point is critical when using Digimat-CAE (see Overview), and especially the "Hybrid" solution method (see subsection Hybrid Solution). We also recommend (if possible) to make these outputs evolve linearly with stress or strain.

For which configurations can I use internal history variables?

Failure (internal) history variables are only available for standard failure criteria without progressive failure; they can only be used in Digimat-MF or Digimat-CAE with Micro solution method.

First Pseudo-grain Failure Model

The main difficulty in using the FPGF scheme to apply failure indicators is to identify the correct FPGF parameters to use.

There are no perfect set of values for those 2 FPGF variables. However, experience has shown that it is preferable to use a **critical fraction of failed pseudo-grains** that ranges from 0.5 to 0.9. If its value is too low it tends to cause an early propagation of the failure in the RVE which is not necessarily desired. On the contrary, if its value is set to 1, there is a little chance that few pseudo-grains never break, or break very lately in the analysis, because their fiber orientations are not in a favorable direction with respect to the load applied.

In general, we recommend to start with the following set of values and to optimize the parameters from that starting point: (FW,CF) = (0.75,0.75). These values are **not** guaranteed to work perfectly well in all cases and must only be considered as good practice values.

The global axis system should never be needed when using the FPGF model. The local axis system based on the orientation of the fibers should be used most of the times. The only case in which the local axis system might not be adapted is for FPGF failure indicators applied on the matrix, which works better with the principal axis system of the evaluated term (stress or strain).

The FPGF driven element deletion happens entirely *independently* of element deletion driven by other standard (non-FPGF) failure indicators. The standard and the FPGF failure mechanisms can be used at the same time and will be taken into account separately.

It is possible to apply FPGF failure indicators to a n-phases composite (n>2), but the FPGF scheme will be applied to **only one** combination of matrix and fiber inclusions with orientation tensor. The name of the inclusion phase that is used in the FPGF scheme is printed in the *.log file, see below.

When using FPGF, the number of pseudo-grains is based on the number of angle increments (see section Orientation).

The maximum number is 173 for a 3D distribution, which is equivalent to 16 pseudo-grains for a 2D distribution. The minimum number of pseudo-grains is 29 for 3D and 6 for 2D distributions. A message will be printed in the *.log file if these bounds are not respected. Using more pseudo-grains will increase the accuracy of the simulation, but will also increase the CPU time. A suggested number of angle increments, for a good compromise between accuracy and time computation on a Digimat-CAE analysis, is 12.

Always check the `*.log` file for FPGF messages. If the FPGF failure indicators are assigned well, the following messages are printed:

```
# DIGIMAT INFO: Checking FPGF inclusion phase definition ...
# DIGIMAT INFO: FPGF applied only for inclusion phase name = GlassFiber
# DIGIMAT INFO: FPGF First Pseudo Grain Failure model = on
```

Note that the second line prints the name of the inclusion phase that, together with the matrix, makes up the pseudo-grains.

Progressive Failure Model

This section intends to provide guidelines for the definition and usage of the progressive failure capabilities in Digimat-MF and for coupled analysis.

Setting Meaningful Parameters for a Progressive Failure Criterion

There are two steps to define a progressive failure model.

1. Set the strength parameters and, most importantly, the damage evolution law. In the following, the strength and damage evolution parameters will be evaluated for of a UD ply, loaded in the fibers' direction (i.e., we will focus on X_t and the evolution of D_{11}), by considering the typical stress-strain curves given if Figure 13-6 (although these curves are much smoother than what could be observed in reality). We assume a Hashin 2D failure criterion with the *linear formulation*, which largely simplifies the interpretation of stress, strain and failure indicators.

   The longitudinal tensile strength X_t corresponds to the stress σ_{ini} at which damage initiates, i.e., where material response becomes non-linear (as we consider an elastic behaviour before damage initiation). It is important to notice that this tensile strength does not always correspond to the peak stress on the curve (σ_{peak}); this is especially true when an exponential damage evolution law is used.

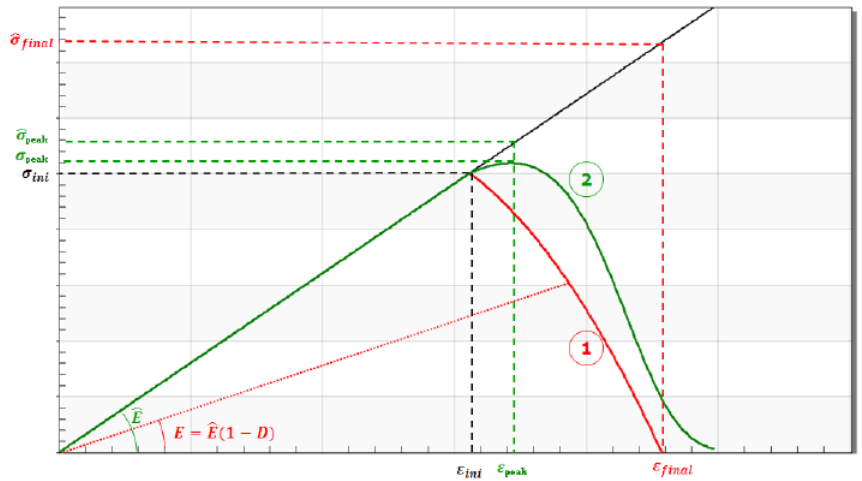


Figure 13-6 Typical stress-strain curves to parametrize progressive failure.

2. Define and parametrize a damage evolution law, among those implemented in Digimat-MF (see subsection Damage Evolution Laws), in order to best-fit the experimental stress-strain curve. The type of evolution law to use must be intuited from the shape of the stress-strain curve, and with the help of the static plots displayed in the Progressive failure tab of the Digimat-MF GUI.

   Remark: The evolution of damage with strain can easily be estimated through the ratio of the secant modulus E, and the original elastic modulus $\hat{E} = E_0$.

The damage evolution laws given in Digimat-MF may look complex at first glance, but reduce to much simpler expressions when some parameters are left to their default value:

- f_{min} should be left to 1.0, according to the definition of X_t given above.

- D_{max} should be left to 1 in the damage law definition. if it is necessary to limit the damage variables, the **damage threshold** progressive failure control can be used instead.
- α Should be left to 1 (for Power and Exponential damage evolution laws).

Then, the damage evolution laws take very simple forms:

- The **instantaneous** damage evolution law triggers a complete and immediate damage, and requires no additional parameter to be set.
- The **power-law** damage evolution law reduces to a linear shape:

$$D = \frac{f - 1}{f_{max} - 1} \tag{13-2}$$

and the stress-strain curve has typically a parabolic shape when the failure indicator reaches 1. The f_{max} parameter can be easily interpreted from Figure 13-6: $f_{max} = \hat{\sigma}_{final} / \sigma_{ini}$. A similar damage evolution law is implemented in ANSYS, and is also discussed in the original paper from Matzenmiller et al. (1995).

Note: An alternative is to set $f_{min} = 0$ and $f_{max} = 1$, and to configure a power-law damage evolution through the α parameter.

Note: A second alternative is to set $\alpha = -1$, which results in an equivalent damage evolution as the linear softening law.

- The **linear softening** damage evolution law reduces to a simpler shape:

$$D = \frac{f_{max}}{f} \times \frac{f - 1}{f_{max} - 1} \tag{13-3}$$

and requires only the f_{max} parameter to be set, using the same expression as for the power-law damage evolution law. It triggers a linear softening of the stress-strain curve towards complete damage.

Note: A similar damage evolution law is implemented in Abaqus.

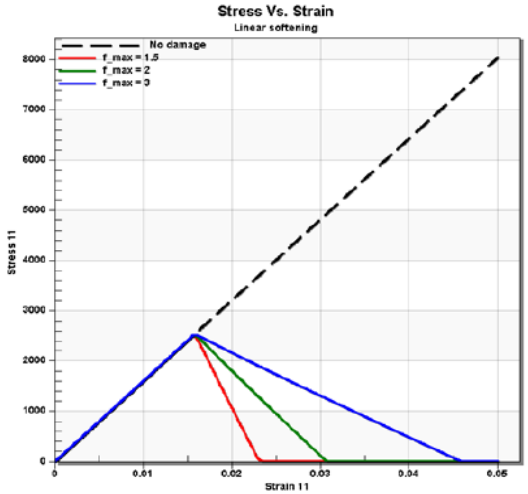


Figure 13-7 Typical stress-strain curve for an linear softening damage evolution law.

- The **exponential** damage evolution law reduces to a simpler shape:

$$D = \left(1 - \exp\left(-\frac{f^{\beta} - 1}{e\beta}\right)\right) \tag{13-4}$$

and requires only the β parameter to be set. It triggers a Weibull-shaped softening of the stress-strain curve towards complete damage.

Note: An alternative is to set $\alpha = e$ and $\beta = 1/e$, which results in a damage evolution law very similar to the one implemented in Marc:

$$D = D_{max} \times (1 - \exp(f_{min} - 1)) \tag{13-5}$$

Remark: Another alternative is to set $f_{min} = 0$ and $= 1$, which results in the following relations, discussed in the original paper from Matzenmiller et al. (1995):

$$D = \left(1 - \exp\left(-\frac{f^{\beta}}{e\beta}\right)\right) \text{ and } \sigma = \exp\left(-\frac{(\hat{\sigma}/X)^{\beta}}{e\beta}\right) \times \hat{\sigma} \tag{13-6}$$

The stress-strain curve then takes the shape of a Weibull function, and the parameter can be estimated from the position of the maximum stress:

$$\beta = m = \frac{1}{\ln(\hat{\sigma}_{peak}/X)} \tag{13-7}$$

The maximum stress value is slightly influenced by the value of the parameter.

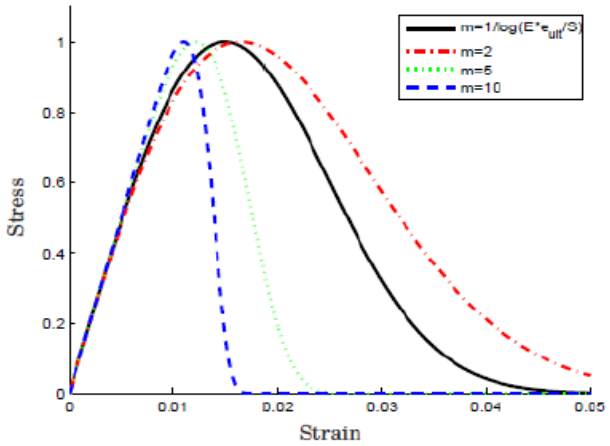


Figure 13-8 Typical stress-strain curve for an exponential damage evolution law (from Gauthier, (2010)).

Loadings

Thermo-mechanical

Over large ranges of temperature, material properties can usually vary quite a lot. This is particularly true for materials presenting transition temperatures within the range of simulated temperatures. It is thus advised to the user to use temperature-dependent properties in these situations.

Reverse Engineering of CTE

The goal of this document is to explain the way Coefficient of Thermal Expansion (CTE, α) are computed in Digimat and how to compare experimental and predicted CTEs in order to perform reverse engineering of the matrix CTE from the composite CTE. This document starts with a recall of the CTE definition and highlights the influence of the reference temperature.

Coefficient of Thermal Expansion

By definition, the CTE are the coefficients of proportionality between the thermal strain experienced by a material and the temperature variation. The CTE are measured by doing DMA/TMA test from T_{min} to T_{max}. The results of these tests are the probe position L function of the temperature. Thermal strain can directly be computed from DMA/TMA test by the following relation:

$$\varepsilon_{ii}^{th}(T) = \frac{(L(T) - L(T_{ref}))}{L(T_{min})} \tag{13-8}$$

where ε_{ii}^{th} is the *ij*-th component of the thermal strain tensor. The secant CTE can then directly be computed from the thermal strain with following relation:

$$\alpha_{ii}(T) = \frac{\varepsilon_{ii}^{th}(T)}{(T - T_{ref})} \tag{13-9}$$

where α_{ii} is the *ij*-th component of the CTE. In Equation and Equation , T_{ref} refers to the reference temperature. It is important to note that both the thermal strain and the CTE are function of the choice of the reference temperature. For $T = T_{ref}$, the secant CTE is defined by the limit of Equation which hence correspond to the tangent CTE at this particular temperature.

Reference Temperature

The reference temperature is chosen as the temperature where the hypothesis is made that there is no thermal strain. Typically, the reference temperature is the temperature of storage prior to use. It is indeed expected that after several days of storage, internal strain will be relaxed in the material. For OEM that cannot know the temperature of storage of their customers, we advise them to

- use $T_{ref} = 23°C$ as default (so only one material file has to be provided).
- generate different material files for different T_{ref} (as well as different CTE as it depends over T_{ref}). So for example, customers who want to perform warpage simulation directly after unmolding should use as reference temperature a T_{ref} close to the molding temperature.

Use of CTE in Digimat

In Digimat, the per-phase relation between the thermal strain, the temperature and the CTE at a given temperature is defined by the following relation:

$$\varepsilon_{th}(T) = \beta(T)(T - T_{ref}) - \alpha(T_{init})(T_{init} - T_{ref}) \tag{13-10}$$

T and T_{init} refer to the current and initial temperatures.

In Digimat, Equation is satisfied at the composite level only with the Hybrid solution. With heterogeneous systems that exhibit temperature-dependent stiffness, simply imposing Equation in the phases does not guarantee to have the same form at the composite level. It is therefore highly recommended to use the reference temperature as initial temperature in Digimat-MF and with the Micro solution in Digimat-CAE. There is no such limitation with the Hybrid solution. The

framework presented above deals with secant CTE. Therefore, the input required by Digimat and the output it provides in the GUI or in the `.eng` files are secant CTE.

Computing Secant CTE from Tangent CTE

It often happens that the results stored from a DMA/TMA test are the tangent CTE. In this case secant CTE can be obtained by computing incrementally the thermal strain from the tangent CTE α_{tgt} with Equation

$$\varepsilon_{ii}^{th}(T_j + 1) = \varepsilon_{ii}^{th}(T_j) + \frac{(\alpha_{ii}^{tgt}(T_j + 1) + \alpha_{ii}^{tgt}(T_j))(T_{j+1} - T_j)}{2} \quad (13\text{-}11)$$

The starting point is $\varepsilon_{th}^{ii}(T_0) = \varepsilon_{th}^{ii}(T_{ref}) = 0$. You therefore need to perform the operation from T_{ref} to T_{min} and from T_{ref} to T_{max} (in the general case for which $T_{min} \le T_{ref} \le T_{max}$). The secant CTE can then be computed from Equation .

Reverse Engineering of Matrix CTE from Composite CTE

Once secant CTE has been obtained at the composite level, the last step is to identify CTE at the matrix level through manual reverse engineering. There are two approaches to identify matrix CTE from composite CTE. The first approach consists in running two thermo-mechanical analyses (assuming $T_{min} \le T_{ref} \le T_{max}$) in Digimat-MF with the following loading conditions

- $T_{init} = T_{ref}$ and $T_{peak} = T_{min}$
- $T_{init} = T_{ref}$ and $T_{peak} = T_{max}$

The mechanical loading has no influence over the results but must be imposed in default global axis system (i.e. Theta=90° and Phi= 0°). Once a thermo-mechanical analysis has been run, you can compare the various predicted CTE outputted in the `.eng` files with the measured CTE. The second approach consists in running an isothermal thermo-mechanical analysis for each temperature at which you have a composite CTE. Once a thermo-mechanical analysis has been run, you can compare the predicted CTE, shown in the **Global Axes** tab of the Stiffness item in the Digimat-MF GUI, with the measured CTE. In both approaches, if predicted and measured CTE are equivalent, your reverse engineering is finished. In the opposite case, you have to rerun your analyses with new guess of the matrix CTE.

14 Known Limitations

- Graphical User Interface
- Homogenization
- Materials
- Microstructure
- RVE
- Failure
- Loadings
- Output

Graphical User Interface

Local Axes Definition

- No error message is issued if the specified local axes are not orthogonal.

Function

- Function with dimension larger than 1 can only be used with fatigue mean-stress sensitivity or with thermo-strain rate elasto-plasticity materials with a dimension of 2.

Homogenization

Double Inclusion

The double inclusion method is known to overestimate the material behavior in the non-linear regime.

Second Order

Transversely isotropic fibers are not officially supported when orientation is defined by an orientation tensor.

Multi-step Scheme

- Analysis with voids are not supported with the multi-step scheme.
- It is known that Matrix + hard reinforcement + soft reinforcement is yielding a wrong result. It is advised to use the Multi-level Scheme in that scenario.

Multi-level Scheme

For linear composites, the multi-level is only available with the combination of:

- an elastic matrix or a thermo-elastic matrix

  with:

  - one elastic inclusion phase with coatings,
  - one elastic inclusion phase with clustering,
  - one elastic inclusion phase together with one void phase,
  - N elastic inclusions phases for which there is at most one phase having
    - aspect ratio larger than one

- orientation which is not fixed orientation with theta=90 and phi=0.

For nonlinear composites, the multi-level is only available with the combination of

- elastoplastic, elastoviscoplastic matrix

  with:

  - one elastic inclusion phase together with one void phase
- elastoplastic matrix

  with:

  - one elastic inclusion phase with coatings
  - one elastic inclusion phase with clustering

Materials

Isotropic Extraction Methods

- The general method is available for all the (thermo-)(visco)plasticity formulations available in Digimat:
  - J_2-plasticity
  - J_2-plasticity with Lemaître-Chaboche damage model
  - Generalized Drucker-Prager plasticity
- The spectral and modified spectral methods are not supported with Lemaître-Chaboche damage model.
- The spectral and modified spectral methods are not supported with the interaction law scheme method.

Linear (thermo)-elasticity

In the inclusion phase, the use of elastic materials is limited to isotropic and transversely isotropic properties.

Elastoplasticity

- The J_2-plasticity model is pressure independent. To account for pressure dependence in the plastic regime, the Drucker-Prager plasticity model can be used.
- In composite analyses involving homogenization, the J_2-plasticity constitutive model can only be combined with the following material models:
  - elastic material
  - elasto-plastic material with fixed orientation for fibers

    - voids
- The incrementally rigid phase behavior is only supported for an elastic inclusion phase in combination with an elasto-plastic matrix.
- The discrete affine linearization method and the interaction law scheme are not supported.
- Second order homogenization scheme with transversely isotropic reinforcements only yields approximate results because the latter are assumed elastic in a step of the homogenization scheme.
- Swift and piecewise hardening laws are restricted to monophase matrix or monophase metal.
- Swift and piecewise hardening laws are not supported with homogenization.

Elastoplasticity: Generalized Drucker-Prager

- Only elastic, elasto-plastic and elasto-viscoplastic inclusions can be used as reinforcements.
- Incrementally rigid formulation and rigid formulation are not available as phase behavior.
- The discrete affine linearization method and the interaction law scheme are not supported.
- Coatings are not supported.
- Clustering is not supported.
- The model is not supported in thermo-mechanical analysis.
- Swift and piecewise hardening laws are not supported.

Hill Plasticity

- Hill plasticity is restricted to monophase matrix or monophase metal.
- Hill plasticity is not supported with homogenization.

Elastoplasticity: Damage Material

- Only isotropic hardening is supported.
- Homogenization is limited to 2-phases when the matrix material behaves according to this model.
- The incrementally rigid phase behavior is not supported.
- Coatings are not supported.
- Clustering is not supported.

- The discrete affine linearization method and the interaction law scheme are not supported.

Thermo-elastoplasticity

- Transversely isotropic fibers are not officially supported when orientation is defined by an orientation tensor.
- The kinematic hardening (Chaboche) model is not available.
- The multi-level method is not available.
  - It implies that for thermo-elastoplastic material containing one inclusion phase and one void phase, only the multi-step method is available. It is reminded that this method is less suited for multi-phase material with void.
- The thermo-elastoplastic model is pressure independent.
- In composite analyses involving homogenization, the thermo-elastoplastic constitutive model can only be combined with the following material models:
  - thermo-elastic material
  - thermo-elasto-plastic material with fixed orientation for fibers
  - voids
- Coatings are not supported.
- Clustering is not supported.
- The incrementally rigid phase behavior is only supported for the combination of an elasto-plastic matriphase with an elastic one.
- The reference temperature must be the same for each material of the RVE.
- The homogenization procedure of composite materials in combination with thermo-dependent material parameters is known to show some difference in the thermo-elastic regime with respect to the same composite modeled by thermo-elastic material models. This is due to the difference of the homogenization procedure implemented for linear and nonlinear materials.
- The discrete affine linearization method and the interaction law scheme are not supported.
- Coefficients of thermal expansion are only available at the initial time with isotropic materials
- Coefficients of thermal expansion are not available as soon as one material is not isotropic

Elasto-viscoplasticity

- The second order homogenization method is only available with the incremental linearization method.

- Coatings are not supported with elasto-viscoplastic materials.
- Clustering is not supported with elasto-viscoplastic materials.
- N-phase composite homogenization is limited to composites with an elasto-viscoplastic matrix phase and elastic inclusion phases.

Thermo-elasto-viscoplasticity

- The second order homogenization method is not available.
- The multi-level method is not available.
- N-phase composite homogenization is limited to composites with an thermo-elasto-viscoplastic matrix phase and thermo-elastic inclusion phases.
- Coatings are not supported with thermo-elasto-viscoplastic materials.
- Clustering is not supported with thermo-elasto-viscoplastic materials.
- The reference temperature must be the same for each material of the RVE.
- The homogenization procedure of composite materials in combination with thermo-dependent material parameters is known to show some difference in the thermo-elastic regime with respect to the same composite modeled by thermo-elastic material models. This is due to the difference of the homogenization procedure implemented for linear and nonlinear materials.
- Transversely isotropic fibers are not officially supported when orientation is defined by an orientation tensor.
- Coefficients of thermal expansion are only available at the initial time with isotropic materials.
- Coefficients of thermal expansion are not available as soon as one material is not isotropic.

Viscoelasticity

- The viscoelastic model is obviously limited to the elastic (small strains) zone of deformation.
- Since $G(t)$ and $K(t)$ must be strictly positive, it is also limited by the fact that it cannot model a softening behavior showing a negative stiffness.
- Coatings are not supported with viscoelasticity materials.
- Clustering is not supported with viscoelasticity materials.
- The viscoelastic model is not supported with:
    - The second order homogenization method;
    - The multi-level method;
    - The discrete affine linearization method;

- The interaction law scheme;
- Initial stresses in input of FE analyses coupled with CAE codes.

Thermo-viscoelasticity

- The thermo-viscoelastic model is obviously limited to the elastic (small strain) zone of deformation.
- Only linear and isotropic thermo-viscoelasticity is available in Digimat:
  - Nonlinear thermo-viscoelasticity is not available
  - Non isotropic definition for the thermal strain is not available
- Since $G_R(t)$ and $K_R(t)$ must be strictly positive, it is also limited by the fact that it cannot model a softening behavior showing a negative stiffness.
- Coatings are not supported with thermo-viscoelastic materials.
- Clustering is not supported with thermo-viscoelastic materials.
- Thermo-viscoelastic model is not supported with
  - The second order homogenization method;
  - The multi-level method;
  - The discrete affine linearization method;
  - The interaction law scheme;
  - Initial stresses in input of FE analyses coupled with CAE codes.

Viscoelasto-viscoplasticity

- Since $G(t)$ and $K(t)$ must be strictly positive, it is also limited by the fact that it cannot model a softening behavior showing a negative stiffness.
- The viscoelastic-viscoplastic model is *not* supported with
  - The second order homogenization method
  - The multi-level homogenization method
- Coatings are not supported with viscoelasto-viscoplastic materials.
- Clustering is not supported with viscoelasto-viscoplastic materials.
- In some specific cases, the homogenization procedure of composite materials involving at least one phase modeled with the VEVP material model may show a light sensitivity to the time step used for the computations.
- The homogenization procedure of composite materials involving at least one phase modeled with the VEVP material model with very high yield stress is known to show some difference with respect to the same composite but involving VE material model. This is due to the difference of the homogenization procedure used to compute the macroscopic behavior of such composite:

    - For VEVP Matrix Composite, the homogenization procedure is done in the time space.
    - For VE Matrix Composite, the homogenization procedure is done in the Laplace-Carson space.
- The interaction law scheme is not supported.
- For some set of parameters unexpected behavior might occur (in this case, please contact support at digimat.support@hexagon.com).

Strain Rate Elastoplasticity

- As strain rate elastoplastic materials are defined at constant strain rates, creep and relaxation loadings are not supported.
- The kinematic hardening (Chaboche) model is not available.
- Lemaitre-Chaboche damage model is not available.
- The discrete affine linearization method and the interaction law scheme are not supported.
- N-phase composite homogenization is limited to composites with an strain rate elastoplastic matrix phase and elastic inclusion phases.

(Thermo)-hyperelasticity

- Only the penalty method is available to enforce the incompressibility constraint in coupled finite element analyses: the augmented Lagrangian method is not available.
- Hyperelastic materials can be used at the matrix level or at the inclusion phase level. It can only be mixed with a hyperelastic material or with elastic or rigid inclusions, and with a Leonov-EGP material.
- Only isotropic modeling is available.
- Coatings are not supported.
- Clustering is not supported.
- N-phase composites are supported provided the matrix phase is modeled with a hyperelastic model. The number of phases is limited to 2 if the matrix is modeled using a linear elastic material with a hyperelastic inclusion phase.

Leonov-EGP

- Second order homogenization scheme and discrete affine method are not supported.
- Multi-level homogenization scheme is not supported.
- Coatings are not supported.
- Clustering is not supported.

- Elastic and rigid inclusions are not supported: only hyperelastic or Leonov-EGP type inclusions are available.
- The usage of this material model in coupled simulations is limited to the Digimat-CAE/Abaqus interface.
- The discrete affine linearization method and the interaction law scheme are not supported.

Fourier Model - Thermal Conductivity

- The Fourier model is a linear model used for static analyses.
- Clustering is not supported.
- Orthotropic materials are not supported at inclusions and coatings level.

Ohm Model - Electrical Conductivity

- The Ohm model is a linear model for static analyses.
- Clustering is not supported.
- Orthotropic materials are not supported at inclusions and coatings level.

Coupled Thermal Mechanical Materials

- Only homogeneous material, UD, basic woven, SMC and chopped fiber with orientation tensor can be supported.
- Void inclusion are not supported.
- Custom output are not supported.
- Orthotropic symmetry is not supported.
- Double inclusion is not supported.
- Log message is reduced.

Microstructure

Generic

- The **incrementally rigid** inclusion behavior is **valid only for elastic inclusions** embedded in an
    - Elasto-plastic matrix.
    - Elasto-viscoplastic matrix.

- The **rigid** inclusion behavior is **valid only for elastic inclusions** embedded in a Hyperelastic matrix.
- Only one inclusion phase per analysis can be characterized with an aspect ratio distribution.
- Coatings are supported with
  - Elastic and elasto-plastic materials
  - Viscoelastic materials:

  if surrounding elastic inclusions embedded in an elastic or a viscoelastic matrix.
- Clustering is only supported for elastic inclusions embedded in an elastic or elasto-plastic matrix.

Other

- Fabrics:
  - Basic and advanced yarns are only supported for a Mechanical and Thermo-Mechanical analysis. Thermal and electrical analyses are not supported.
  - Advanced yarns are only supported for elastic inclusions embedded in an elastic matrix.
- Lattice
  - Only non-reinforced lattice are supported.
  - Only elastic and J_2-plasticity matrix are supported.
  - Failure is only supported at the composite level.
  - Only the spectral isotropic extraction method is available.
- Sheet molding compound.
  - Only elastic, thermo-elastic, visco-elastic and thermo-visco-elastic materials are supported.

RVE

- For multi-layer RVEs
  - The matrix phase of each layer must be defined with the same material.
  - Only 2-phase materials can be defined in each layer, which implies that the multi-level homogenization scheme, coatings phases, clustering and the usage of aspect ratio distributions are not available.
  - Finite strain materials (Leonov-EGP and hyperelastic models) are not supported.
  - The Lemaître-Chaboche damage model cannot be used.
  - Kinematic hardening cannot be used.

    - Percolation of the inclusion phase cannot be accounted for.
    - `BIAXIAL1_23` loading cannot be applied.
    - Coatings and clustering are not supported.
- Laminate materials cannot be used in a coupled FE analysis with Digimat-CAE.
- For woven with basic yarns, only 2D woven are supported.
- For woven and braided microstructures with advanced yarns:
    - Only mechanical analyses can be performed (following the limitations on yarns).
    - Failure can only be defined at matrix, yarn and composite level, not at the level of the matrix and fibers inside the yarns.
    - Output are only available at matrix, yarn and composite level, not at the level of the matrix and fibers inside the yarns.
    - The use of different yarn in a single direction is not supported.

Failure

General Limitations

- It is not allowed to use both FPGF and Standard failure mechanisms at the same time.
- A failure criterion can only be assigned to the composite level:
    - For multi-layer materials with at least one viscoelastic material,
    - For composites with more than 2 phases with at least one viscoelastic material.
- The Accumulated plastic strain criterion cannot be used at the macroscopic (Composite) level, unless homogenization is switched off.
- User-defines failure indicators using history variables cannot be used for the following configurations in Digimat-MF:
    - FPGF failure criteria,
    - progressive failure criteria
    - when the number of history variable is higher than 99 (for IO formatting reasons).
- It is not allowed to use a failure criterion with SMC materials which have a thermal dependency (when running thermomechanical or coupled thermal mechanical).

Progressive Failure Model

- Progressive failure can only be assigned to a linear elastic material without dependencies and without thermal effects.
- Progressive failure can only be assigned to the composite level.

- A damage model can only be attached to a failure criterion defined in the local axis system, without strain rate dependencies, and using the standard failure mechanism (no FPGF).
- Progressive failure cannot be used with a 2.5D or 3D woven fabric RVE (involving advanced yarns) or a braided fabric RVE.
- Progressive failure cannot be used with a multilayer RVE.
- Progressive failure cannot be used with a stress-based loading.

First Pseudo-grain Failure Model

- The FPGF failure model is not supported for composites defined with fixed fiber orientation.
- FPGF scheme can only be applied to composite materials with a matrix behavior which is
  - Elasto-plastic.
  - Elasto-viscoplastic.
  - Viscoelastic-viscoplastic.
- FPGF scheme cannot be applied to composite materials with more than three phases.
  - If there are three phases in the composite, one of the two inclusion phases must be described with an orientation tensor and the other one with a fixed orientation.
- FPGF is activated only if the matrix plastic yield point has been reached. This implies that
  - FPGF is not available for elastic composites (all phases being elastic).
  - The FPGF outputs will remain at 0.0 in the elastic zone, even if failure indicator values are non-zero.
- When the RVE elastically unloads, the elastic slope (tangent stiffness) will be the same as for the first elastic load (i.e., pseudo-grain stiffness contribution removal, if activated, has no effect).
- FPGF is active only for composites that have pseudo-grains, this means composites with a fiber orientation distribution described as
  - Random 2D
  - Random 3D
  - Orientation tensor
- FPGF is not activated for spherical inclusions because there is no notion of orientation for such inclusion shape.
- It is not allowed to apply FPGF failure indicators both on the composite and on the phase at the pseudo-grain level simultaneously.

- In the case of a multilayer analysis, it is not possible to assign different failure criteria from one layer to another. If a failure criterion is applied on the matrix phase at the pseudo-grain level, it will automatically be applied to all the RVE layers.
- It is not allowed to mix both FPGF and standard failures indicators at the same time to reach the break point of the composite material.
    - The material response is influenced exclusively by FPGF even if the standard failure at the macroscopic level or at phase level exceeds 1.
- When using FPGF with a user-defined failure indicator, the "accumulated plastic strain" passed as input to the failure subroutine (for a given pseudo-grain) is the phase-level average, not the pseudo-grain specific value.

Pseudo Grain Fatigue Model

- This model is restricted to non-homogeneous materials.
- This model is restricted to composites made of a (visco)elastic matrix and one (and only one) elastic inclusion phase. Pseudo grain fatigue can be combined with elastoplastic in the graphical interface but this behavior to enable plasticity correction in nCode and CAEfatigue.
- Only the Tsai-Hill 3D transversely isotropic failure criterion can be used to evaluate the lifetime of the RVE.
- A fatigue failure indicator cannot be assigned simultaneously with any other failure indicator.
- The stress loading can only be defined in terms of cyclic history (not monotonic or user-defined).
- Deprecation: In Digimat 2019.0 and later versions, it is not possible anymore to load in the Digimat GUIs an analysis file which contains a fatigue failure indicator with non-fixed orientation types associated to experimental S-N curves input for reverse engineering. The reverse engineering of a fatigue failure indicator is now performed exclusively in Digimat-MX. Please contact your material supplier for an updated version of the analysis file.
- Deprecation: In Digimat 2019.0 and later versions, the way the pseudo-grain fatigue failure criteria is defined and written in the Digimat analysis file (DAF) has been changed. The new version now looks much more similar to the standard Tsai-Hill 3D transversely isotropic stress-based failure indicator. It remains consistent with the theoretical model, and has no significant impact of the results whatsoever.
  When loading such an old-format DAF file in Digimat-MF or Digimat-CAE, the fatigue failure indicator will be converted to the new format, unless the file is encrypted; in such case, please contact your material supplier for an updated version.

Modified Gerber Model

- The modified Gerber model can only be used in Digimat in conjunction with a static failure criterion applied at the pseudo grain level.
- The static failure indicator can be used only with linear formulation option.
- Multi-layer analysis is not supported for the modified Gerber model in Digimat-MF.
- Fatigue loading "Number of cycles" can only be used with a stress-based Tsai-Hill 3D TI failure indicator.
- Encrypted files are not supported in Digimat-MF for the modified Gerber fatigue model.
- Accumulated plastic strain failure criterion is not supported in combination with modified Gerber fatigue model.
- Standard failure indicators assigned at composite level are not supported in combination with modified Gerber fatigue model.

Energy-based Criterion

- The energy-based criterion can only be used in Digimat MF in conjunction with a strain rate elastoplastic or viscoelastic-viscoplastic material models for the matrix phase.
- Fatigue loading "Number of cycles" is not supported for the energy-based fatigue criterion.
- Multi-layer analysis is not supported for the energy-based fatigue criterion in Digimat-MF.
- The energy-based criterion cannot be assigned simultaneously with any other failure indicator.

Matrix Damage Fatigue Model

- This model is dedicated to two-phase composites involving inclusions with fixed orientations and to fabric microstructure using basic yarns.
- This stress loading cannot be defined in terms of monotonic history or numbers of cycles range (with cyclic history) with this model.

Loadings

Mechanical

- With strain rate dependent material models the option 'Use quasi-static loading' does not verify after the computation that the loading is really quasi-static.
- Stress type and cyclic history type loadings are not available with finite strain materials (hyperelastic and Leonov-EGP).

- In case of simulation of a uniaxial compression test with finite strain materials, the peak strain cannot be lower than -1.
- Stress type loadings are only available with elastic and viscoelastic material models.
- The use of `GENERAL_3D` loading, with a monotonic or a user-defined history loading for incompressible hyperelastic materials can lead to a non respect of the incompressibility condition between two loading state.
- Harmonic stress and strain loading types are only available with elastic and viscoelastic material models.

Output

General

- For multi-layer materials, macroscopic results per layer are not available.
- Per-phase results are not available:
  - For multi-layer materials with at least one viscoelastic material,
  - For composites with more than 2 phases with at least one viscoelastic material.

Failure Envelope

The following limitations apply to the failure envelope capability:

- One or several failure indicator(s) must be assigned in the "Failure assignment tab".
- The analysis file must not be crypted.
- The analysis type must mechanical or thermo-mechanical.
- The RVE definition can only use a single microstructure (but can be a classical or a multilayer RVE).
- The microstructure can only be a single-phase, bi-phase or a basic woven; N-phase and advanced wovens are not supported.
- When the microstructure is mono-phase (or if homogenization is switched off), the following limitations apply:
  - Multilayer RVE cannot be used;
  - Failure criteria must be assigned at the composite level (even though the composite and matrix levels are equivalent);
  - The "E22 vs. 2×E12" and "S22 vs. S12" loadings cannot be used.
- The capability cannot be used with finite strains, or when a hyperelastic material is used.

Carpet Plot

The following limitations apply to the carpet plot capability:

- The analysis file must not be encrypted.
- The analysis type must mechanical or thermo-mechanical.
- The RVE definition can only use a single microstructure.
- The microstructure can only be a bi-phase UD (with continuous fibers) or a basic woven; mono-phase, N-phase and advanced wovens are not supported.
- Homogenization must be switched on.
- The capability cannot be used with finite strains, or when a hyperelastic material is used.
- The capability cannot be used when a progressive failure criterion is active (due to usage limitations for progressive failure with multilayer RVE).
- The failure output is only available with Standard failure criteria applied at micro (phase) level, as:
    - Standard failure criteria at Composite (macroscopic) level are not meaningful, as they do not take into account layer orientations;
    - FPGF criteria are not applicable to continuous fibers;
    - Progressive failure criteria are not applicable when the RVE is multilayer.

Forming Limit Diagram

The following limitations apply to the FLD capability:

- The analysis file must not be crypted.
- The analysis type must mechanical.
- The RVE definition can only use a single metal microstructure.
- The capability can only be used with plasticity or crystal plasticity.
- The capability cannot be used with failure.

Engineering Constants

- The density of the composite is calculated only if the density of each phase is given by the user.
- The heat capacity of the composite is calculated only if the heat capacity of each phase is given by the user.
- The local compliance and the local stiffness matrix are written only for a 2-phase composite or for N-phase composite for which each inclusion phase have exactly the same orientation.

- The macroscopic and local thermal expansion matrix are not available for thermo-viscoelastic material and only at the first increment for thermo-elastoplastic and thermo-elasto-viscoplastic matrix composite. If the first increment is not elastic, the CTE printed is only an approximation.
- The engineering moduli (and also the orthotropic angles) are computed only for a 2-phase composite or for N-phase composite for which each inclusion phase have exactly the same orientation.
- The orthotropic angles are only available if a orthotropic projection is used to compute the engineering constants.

References

1. Y. Benveniste. A new approach to the application of Mori-Tanaka theory in composite materials. Mech. Mater., 6:147–157, 1987.
2. Z. Hashin and S. Shtrikman. A variational approach to the theory of the effective magnetic permeability of multiphase materials. Journal of Applied Physics, 33:3125–3131, 1962.
3. J. Eshelby. The determination of the elastic field of an ellipsoidal inclusion and related problems. Proceedings of the Royal Society of London, pages 376–396, 1957.
4. T. Mori and K. Tanaka. Average stress in the matrix and average elastic energy of materials with misfitting inclusions. Acta Metall. Mater., 21:571–574, 1973.
5. S. Nemat-Nasser and M. Hori. Micromechanics: overall properties of heterogeneous solids. Elsevier Science, 1993.
6. G. Lielens. Micro-macro Modeling of Structured Materials. PhD thesis, Université catholique de Louvain, 1999.
7. J. S. Cintra and C. L. Tucker. Orthotropic closure approximations for flow-induced fiber orientation. Journal of Rheology, 39:1095, 1995.
8. E. Onate and F. Leckie. Representation of mechanical behavior in the presence of changing internal structure. Journal of Applied Mechanics, 55:1–10, 1998.
9. R. A. Schapery. Approximate methods of transform inversion for viscoelastic stress analysis. In Proc. 4th US Nat'l Cong. Appl. Mech, 1962.
10. J. Lemaitre and J.-L. Chaboche. Mechanics of solid materials. Cambridge University Press, 1990.
11. Olivero and Radford. A multiple percolation approach to EMI shielding composites incorporating conductive fillers. Journal of Reinforced Plastics and Composites, 17:674–690, 1998.
12. R. Talreja. A continuum mechanics characterization of damage in composite materials. Proceedings of the Royal Society of London, Series A (Mathematical and Physical Sciences), 399:195–216, 1985.
13. L. Guangyan. Damage progression in open-hole tension composite laminates by the element-failure method. PhD thesis, National university of Singapore, 2007.
14. A. Matzenmiller, J. Lubliner, and R. Taylor. A constitutive model for anisotropic damage in fiber-composites. Mechanics of Materials, 20(12):125–152, 1995.
15. P. Maimí, P. Camanho, J. Mayugo, and C. Dávila. A continuum damage model for composite laminates: Part I — constitutive model. Mechanics of Materials, 39(10):897–908, 2007.
16. Y. Liu. A unified multiaxial fatigue damage model for isotropic and anisotropic materials. International Journal of Fatigue, 29(2), 2007.

17. R. Desmorat, A. Kane, M. Seyedi, and S. J.P. Two scale damage model and related numerical issues for thermo-mechanical high cyclic fatigue. European Journal of Mechanics A/Solids, 26:909–935, 2007.

18. I. Doghri. Mechanics of Deformable Solids - Linear and Nonlinear, Analytical and Computational Aspects. Springer, 2000.

19. Composite materials handbook (CMH-17), volume 3. ASTM International, West Conshohocken, Pennsylvania, USA, 2009.

20. G. Hu, A. A. Tay, Y. Zhang, W. Zhu, and S. Chew. Characterization of viscoelastic behaviour of a molding compound with application to delamination analysis in IC packages. In Electronics Packaging Technology Conference, pages 53–59. IEEE, 2006.

21. L. Gauthier. Modelling of High Velocity Impact on Composite Materials for Airframe Structures Application. PhD thesis, Faculté des sciences et de génie, Université de Laval, Québec, 2010.

22. M. Berveiller and A. Zaoui. An extension of the self-consistent scheme to plastically-flowing polycrystals. J. Mech. Phys. Solids, 26:325–344, 1979.

23. E. Kröner. On the plastic deformation of polycrystals. Acta Metall., 9:155–161, 1961.

24. S. Lee, C. M.T., and L. H.S. Kinetic model for the curing reaction of a tetraglycidyl diamino diphenyl methane/diamino diphenyl sulfone (tgddm/dds) epoxy resin system. Polym. Eng. Sci., 32(15):1037–1046, 1992.

25. M. Kamal and S. Sourour. Kinetics and thermal characterization of thermoset cure. Polym. Eng. Sci., 13(1): 59–64, 1973.

26. M. Bornert, T. Bretheu, and P. Gilormini. Homogénéisation en mécanique des matériaux. Hermes Science, 2000.

27. A. Johnston. An integrated model of the development of process-induced deformation in autoclave processing of composite structures. PhD thesis, The University of British Columbia, 1997.

28. P. Mallick and Y. Zhou. Effect of mean stress on the stress-controlled fatigue of a short e-glass fiber reinforced polyamide-6,6. International Journal of Fatigue, 26(9):941–946, 2004.

29. Mercier and A. Molinari. Homogenization of elastic-viscoplastic heterogeneous materials: Self-consistent and mori-tanaka schemes. International Journal of Plasticity, 25:1024–1048, 2009.

30. A. Molinari. Averaging models for heterogeneous viscoplastic and elasto-viscoplastic materials. J. Eng. Mater. Technol., 124:62–70, 2002.

31. J. Chaboche, P. Kanouté, and A. Roos. On the capabilities of mean-field approaches for the description of plasticity in metal matrix composites. Int. J. Plasticity, 21:1409–1434, 2005.

32. O. Pierard and I. Doghri. An enhanced affine formulation and the corresponding numerical algorithms for the mean-field homogenization of elasto-viscoplastic composites. International Journal of Plasticity, 22: 131–157, 2006a.

33. O. Pierard and I. Doghri. Study of various estimates of the macroscopic tangent operator in the incremental homogenization of elasto-plastic composites. International Journal of Multiscale Computational Engineering, 4:521–543, 2006b.

34. I. Doghri and A. Ouaar. Homogenization of two-phase elasto-plastic composite materials and structures. study of tangent operators, cyclic plasticity and numerical algorithms. International Journal of Solids and Structures, 40:1681–1712, 2003.

35. Drucker and W. Prager. Soil mechanics and plastic analysis or limited design. Quarterly Applied Mathematics, 10(2):157–165, 1952.

36. I. Doghri, L. Adam, and N. Bilger. Mean-field homogenization of elasto-viscoplastic composites based on a general incrementally affine linearization method. International Journal of Plasticity, 26:219–238, 2010.

37. J. Simo and R. Taylor. Quasi-incompressible finite elasticity in principal stretches. continuum basis and numerical algorithms. Computer Methods in Applied Mechanics and Engineering, 85:273–310, 1991.

38. Z. Hashin. Fatigue failure criteria for unidirectional fiber composites. Journal of Applied Mechanics, Transactions ASME, 48(4):846–852, 1981.

39. A. P. Vassilopoulos, B. D. Manshadi, and T. Keller. Influence of the constant life diagram formulation on the fatigue life prediction of composite materials. International Journal of Fatigue, 32(4):659–669, 2010. ISSN 0142-1123.
doi: http://dx.doi.org/10.1016/j.ijfatigue.2009.09.008.
URL: http://www.sciencedirect.com/science/article/pii/S0142112309002795.

40. J. Lemaitre and I. Doghri. Damage 90: a post processor for crack initiation. Comput. Methods Appl. Engrg., 115:197–232, 1994.

41. Hutchinson, J.W., Neale, K.W., 1977a. Sheet necking II: time-independent behaviour. In: Koistinen, D.P., Wang, N.-M. (Eds.), Mechanics of Sheet Metal Forming: Material Behavior and Deformation Analysis: 1977 GMR Symposium. Plenum Press, pp. 127–153. ISBN: 0-306-40068-5.

42. Marciniak, Z., Kuczynski, K., 1967. Limit strains in the processes of stretch-forming sheet metal. International Journal of Mechanical Sciences 9, 609–620.

43. Wu, P.D. & Neale, K.W. & Van der Giessen, Erik. (1997). On crystal plasticity FLD analysis. Proceedings of the Royal Society A: Mathematical, Physical and Engineering Sciences. 453, 1831-1848

44. L. Toth and P. Van Houtte. Discretization techniques for orientation distribution functions. Textures and Microstructures, 19(4):229–244, 1992.

45. P. Eisenlohr and F. Roters. Selecting a set of discrete orientations for accurate texture reconstruction. Computational Materials Science - COMPUT MATER SCI, 42:670–678, 06 2008. doi: 10.1016/j.commatsci.2007.09.015.

46. B. Miled, I. Doghri, and L. Delannay. Coupled viscoelastic-viscoplastic modeling of isotropic polymers: numerical algorithm and analytical solutions. Computational Methods in Applied Mechanical Engineering, 200: 3381–3394, 2011.

47. R. Hill. (1948). A theory of the yielding and plastic flow of anisotropic metals. Proc. Roy. Soc. London, 193:281–297

48. Ilan RAPHAEL, Nicolas SAINTIER, Helloise ROLLAND, Gille ROBERT, Lucien LAIARINANDRASANA - A mixed strain rate and energy based fatigue criterion for short fiber reinforced thermoplastics - International Journal of Fatigue, Volume 127, p.131-143 - 2019.

49. P. Santharam, Y. Marco, V. Le Saux, M. Le Saux, G. Robert, I. Raoult, C. Guévenoux, D. Taveau, P. Charrier. Fatigue criteria for short fiber-reinforced thermoplastic validated over various fiber orientations, load ratios and environmental conditions, International Journal of Fatigue, Volume 135, 2020, 105574, ISSN 0142-1123.