VA

Digimat 2023.3

VA 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

September 25, 2023

DIGI:V2023.3: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 VA User's Guide

Preface

1 Overview

2 Test Matrix

3 Simulation

4 Allowables

5 File Data Management

6 Command Line

7 Specific Features

8 Known Limitations

References

Preface

About This Guide

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

Purpose of This Guide

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

- An overview of Digimat VA capabilities for performing virtual material testing
- Details of the GUI of Digimat-VA and explain standard usage workflows
- Information to perform post-processing of results and understand allowables
- An overview for you of how to run an analysis in batch mode or on the command-line

The information in this manual is mostly 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 5: File Data Management
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

- Prerequisites
- General Description
- Allowables and Composite Characterization
- Classical Laminate Theory
- Software Usage

Prerequisites

The Microsoft .NET Framework 4.8 or higher is required to use Digimat-VA. For details on how to get it, refer to the *Installation of prerequisites on Windows platforms* in the *Installation and Operations Guide*.

General Description

Digimat-VA is an integrative solution that allows easy and efficient prediction of virtual allowables. It combines multiscale modeling, a flexible failure modeling framework, non-linear finite element analysis, stochastic methods and automated post-processing fully tailored for the simulation and prediction of composite laminate coupon testing. It is representative for standard composite characterization methods. It offers the user to walk a 3-step workflow, from test matrix definition to allowables reporting (see Software Usage).

Allowables and Composite Characterization

Allowables constitute fundamental material properties used for composite structural design. They may involve loading plates for which dimensions and the constitutive material must be specified.They consist in statistically-derived values representative for the behavior of a composite material system in a given structural context. In particular, they quantify the strength of the material as characterized by various coupon tests according to dedicated standard methods. Hence they proceed from the measurement of much more data, e.g., stress-strain curves, which standards do not necessarily recommend to record.

Allowables are collected from tests defined after a structural complexity level and a data application category. Structural complexity levels extend from material constituent to structural component, data application categories from material screening to structural substantiation (see CMH-17).

Covering successive structural complexity levels and data application categories ends up in a so-called building block approach, where more tests are performed for the first structural complexity levels and data application categories than the next. For instance, numerous laminate tests are performed for several material systems in a screening phase, then for the single selected material system in a qualification phase. In such a phase, less structural element, e.g., bearing, tests are performed. In this sense, Digimat-VA enables to simulate laminate and structural element (notched and bearing) tests from lamina data.

Despite their structural nature, notched tests still provide allowables considered as material properties. They characterize the material notch sensitivity, critical for composite structures commonly assembled with fasteners.

- Open hole tests helps characterize the material notch sensitivity. The test consist in uniaxial tensile or compressive tests on coupons made of various layups and drilled with a hole at half their length (Figure 1-1). This hole triggers a stress concentration which initiates failure about the hole. The ratio between the hole diameter and the coupon width is standardized, e.g., to 1/6th according to ASTM D5766 (tension) or 6484 (compression), in order to enable meaningful comparisons. The corresponding strength does not correspond to a local material stress value. Indeed it equals the maximum measured force divided by the **unnotched** cross-sectional area. In particular, it is not representative for the critical stress concentration about the hole.

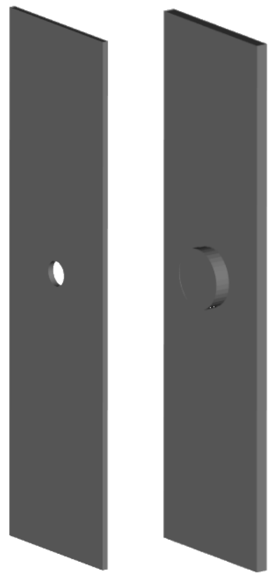


Figure 1-1 Notched tests: open hole on the left or filled hole on the right.

- Filled hole tests globally correspond to open hole ones but, in addition, involve a fastener within the hole. The fastener head and hole may be countersunk or not as described by, e.g., ASTM D6742. Furthermore, the fastener is subjected to a certain tightening torque. This torque induces a preload between the coupon and the fastener in the shape of additional forces which, in terms of stiffness and strength, influence the coupon global response.

Providing first properties of structural nature, bearing tests involve more degrees of freedom. Indeed they characterize the first failure mode of mechanically fastened joints possibly exhibiting very different setups. Hence bearing tests represent the fundamental features of such setups. To this extent, standards such as ASTM D5961 recommend to test simple assemblies of a fixed and a loaded part jointed by at least one fastener (Figure 1-2). These assemblies and associated procedures differ over the following features:

- Single or double shear loading according to the number of parts transmitting the load to the coupon via the fastener;
- The number of composite coupon pieces, 1 or 2 for single shear bearing;
- The number of fasteners, single or double for single shear bearing 2-piece;

- The loading type, tensile or compressive for single shear bearing 2-piece as well as possible stabilization for tensile loading (always used for compressive loading).

They may involve loading plates which dimensions and constitutive material must be specified. They also significantly differ from lower structural complexity level tests on data measurement and calculation aspects. Indeed, the strength relates to the so-called bearing area (and the hole diameter) instead of a cross-sectional area (and the coupon width). In addition, the strain is deduced from the relative displacement of both parts of the assembly normalized by the hole diameter and not the initial distance between the measurement points. Hence the bearing strain adopts larger values than strain collected from lower structural complexity level tests.

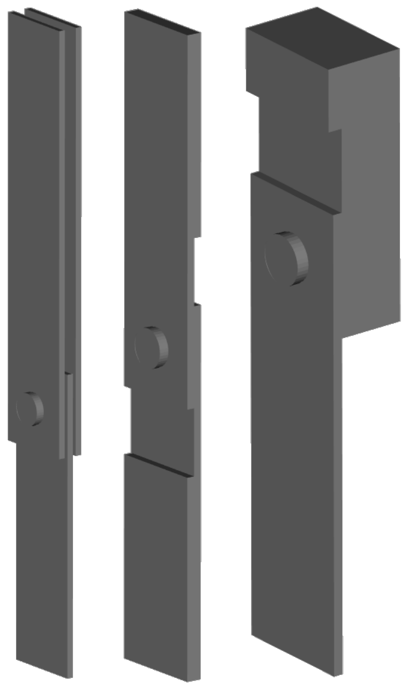


Figure 1-2 Bearing tests – double shear on the left, single shear 2-piece on the middle or single shear 1-piece on the right – characterize the first failure mode of mechanically fastened joints.

Classical Laminate Theory

In addition to the detailed approach based on finite element using material models with progressive failure, Digimat-VA can also provide analytical estimations of stiffness and strength of laminates based on the Classical Laminate Theory (CLT). The CLT is a direct extension of the classical plate theory for isotropic and homogeneous material, as proposed by Kirchhoff-Love. However, the extension of this theory to laminates requires some modifications to take into account the inhomogeneity in thickness direction.

The main assumptions are the following:

- The laminate consists of perfectly bonded layers. In other words, it is equivalent to saying that the displacement components are continuous through the thickness.
- Each lamina is considered to be a homogeneous layer, whose effective properties are known.
- Each lamina is in a plane stress state.
- The individual lamina can be isotropic, orthotropic or transversely isotropic.
- The laminate deforms according to the Kirchhoff - Love assumptions for bending and stretching of thin plates. Those assumptions are:
    - The normals to the mid-plane remain straight and normal to the mid-plane, even after deformation.
    - The thickness of the plate does not change during deformation.

In Digimat-VA, the CLT approach can be used to obtain estimations for:

- Laminate elastic stiffness
- CTE (coefficients of thermal expansion)
- CME (coefficients of moisture expansion)
- Strength (first ply failure)

Software Usage

Digimat-VA working environment is organized around projects. When starting Digimat-VA, the user can either

- Create a new project
- Load a recently saved project
- Load an existing project from the hard drive

A project is defined by a name, a working directory as well as a unit system. Once the definition of the project is done, the user can access the main window of Digimat-VA. The main window of Digimat-VA is divided in three sections as shown in Figure 1-3.

- Upper workflow ribbon
- Working screen
- Lower workflow ribbon

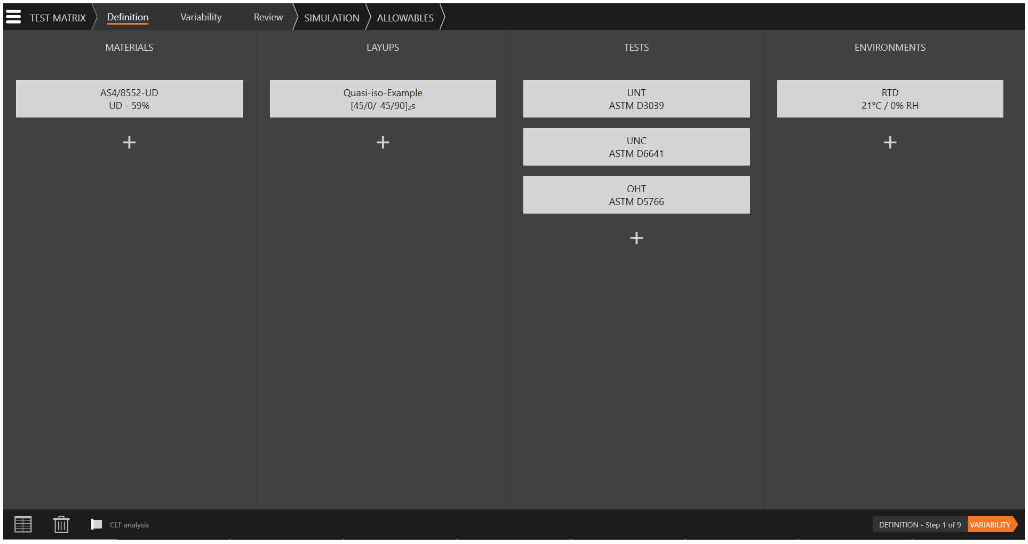


Figure 1-3 Digimat-VA main window.

The lower ribbon acts as a one-way wizard and gives the current step in the workflow as well as access to the next step.The working screen gives access to all information required to work in the current step of the workflow. Each of these screens are described in the following sections.

The upper workflow ribbon is made of three key steps:

- Test Matrix
- Simulation
- Allowables

These three steps highlight the spirit of the software.

Like any physical testing campaign, the first step requires the definition of the test matrix content, based on the required number of materials, layups, coupon tests and environment conditions to be tested.

Once the test matrix is defined, including the number of repetition of each test, it contains all information to be sent to an experimental lab. But obviously, the idea is to provide that test matrix to Digimat-VA, which will turn it into simulations. Simulation requires two inputs, a Digimat material model as well as a finite element model. Those two inputs are created in the Simulation step. They can then be run and monitored until results are obtained.

Once simulations are over, automated post-processing is proposed in the Allowables step. From simulation results, global results such as stress-strain curves, stiffness and strength are extracted. If variability is activated, A and B-basis values are automatically computed. If the user is interested in local results, stress/strain fields and failure pattern visualization are also accessible. Finally, all input and output of a Digimat-VA project can be synthesized in a Word document report, ready to be further detailed or shared.

2 Test Matrix

- General description
- Material Definition
- Layup Definition
- Standard Test Definition
- Environment Conditions
- Test Matrix Edition
- Variability definition

General description

A test matrix is defined after a set of

- materials
- layups
- tests
- environments.

In the test matrix definition screen (see Figure 2-1), each test matrix item is displayed as a tile, which can be activated or deactivated by left clicking on it. When hovering over a tile, three action buttons appear on the right part of the tile, allowing to edit, save to the database (see Database) or remove the selected item from the test matrix definition.

Test matrix items can be added by clicking on the plus icon in each column. Each test matrix item can either be retrieved from the Digimat-VA database or be defined on the fly. For every test matrix item, a specific unit system is always attached (with the exception of layups, which don't have any dimensional parameter). For all dimensional parameters, the unit to be used is clearly indicated. Changing the selected unit system causes a conversion of all dimensional parameters.

In the bottom left corner of the window, three buttons allow to

- Selectively activate/deactivate tests (see Test Matrix Edition)
- Delete all items in the test matrix
- Activate/deactivate the CLT computations (see Classical Laminate Theory)

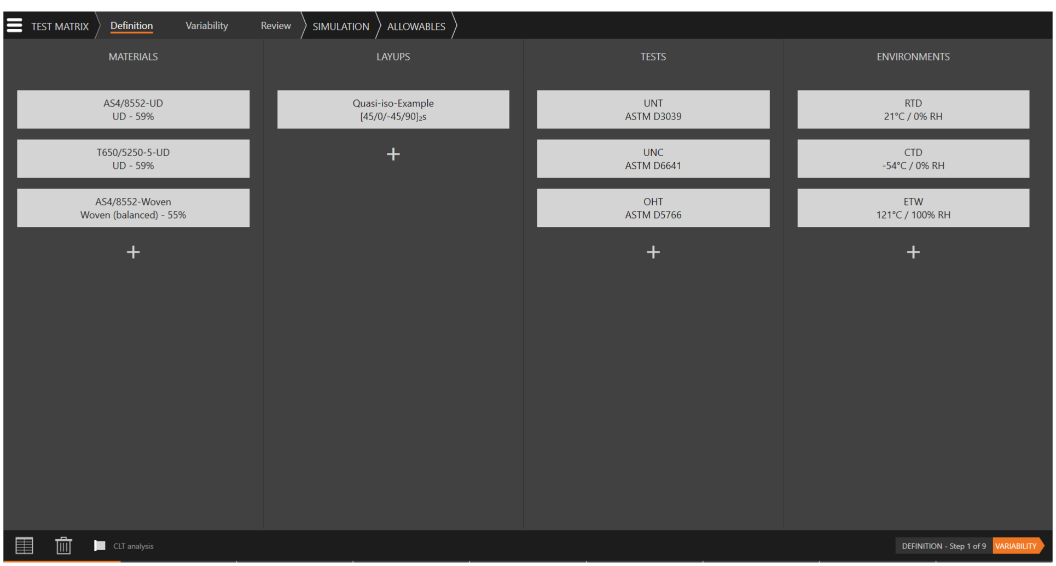


Figure 2-1 Test matrix definition.

Material Definition

The definition of a material (also called system) requires the following data:

- Material type: UD or woven (balanced and unbalanced)
- Matrix name
- Fiber name
- Nominal volume fraction
- Warp weight rate (only for unbalanced woven)
- Weft weight rate (only for unbalanced woven)
- Cured ply thickness

Layup Definition

A layup definition consists of a sequence of plies, each ply having its own orientation. The default behavior in Digimat-VA is to assume that all plies are made of the same system. In that case, no specific system is assigned to a layup: that step will be done when the test matrix is built. Hybrid layers are supported as well. In that case, it is necessary to assign a system for each ply.

The layup definition window is illustrated in Figure 2-2.

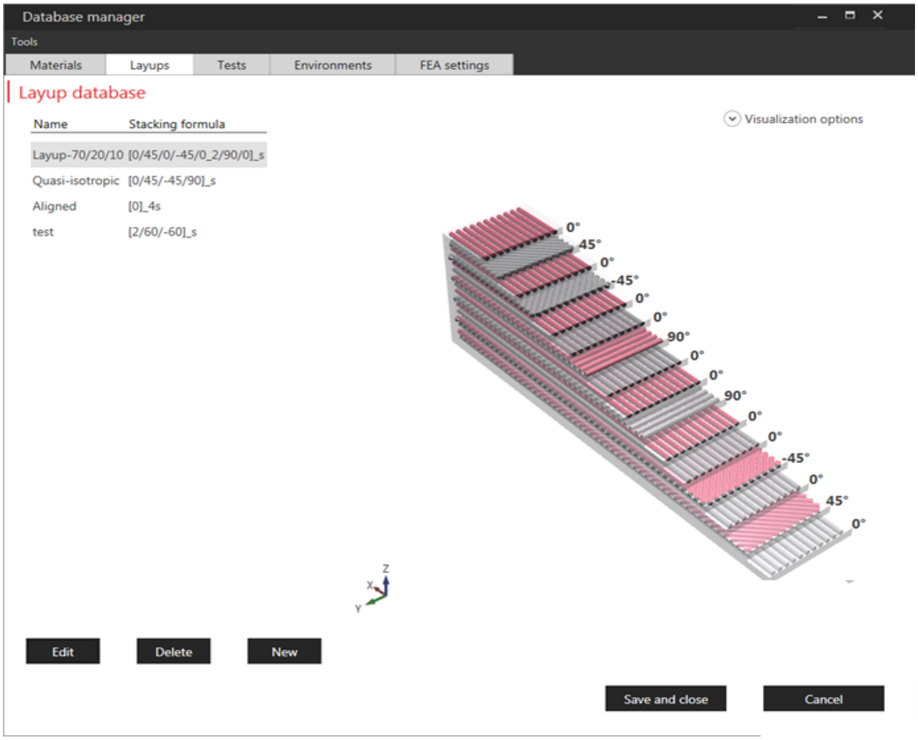


Figure 2-2 New layup definition.

Layups can be built either by directly adding plies and specifying orientation for each of them, or by providing the stacking formula and clicking the **Create** button from stacking formula. The stacking formula must follow this format: **[a/b/c]_xs**

- The first part must be enclosed in square brackets. Inside the square brackets, the orientation of each ply is specified (top to bottom) with respect to the longitudinal (or warp) fiber direction, with / being used as separator. If two or more consecutive plies share the same orientation, they can be condensed with the notation: **y_x**, where y is the orientation and x is the number of repetitions.
- The second part (facultative) must have the format **_xs**, where x is the number of times the sequence specified in the square brackets must be repeated and s indicates that the layup should be made symmetric (i.e. that only half of the plies are specified).

Carpet Plot Layup

A second type of more specific layup also exists. It is called the carpet plot layup. A carpet plot is a plot illustrating the variation of a laminate stiffness or strength for varying layup. It involves a layup with three different ply orientations, where the proportion of each of the three different ply orientation varies (from 0 to 100 percent). An example of such a plot is shown in Figure 2-3.

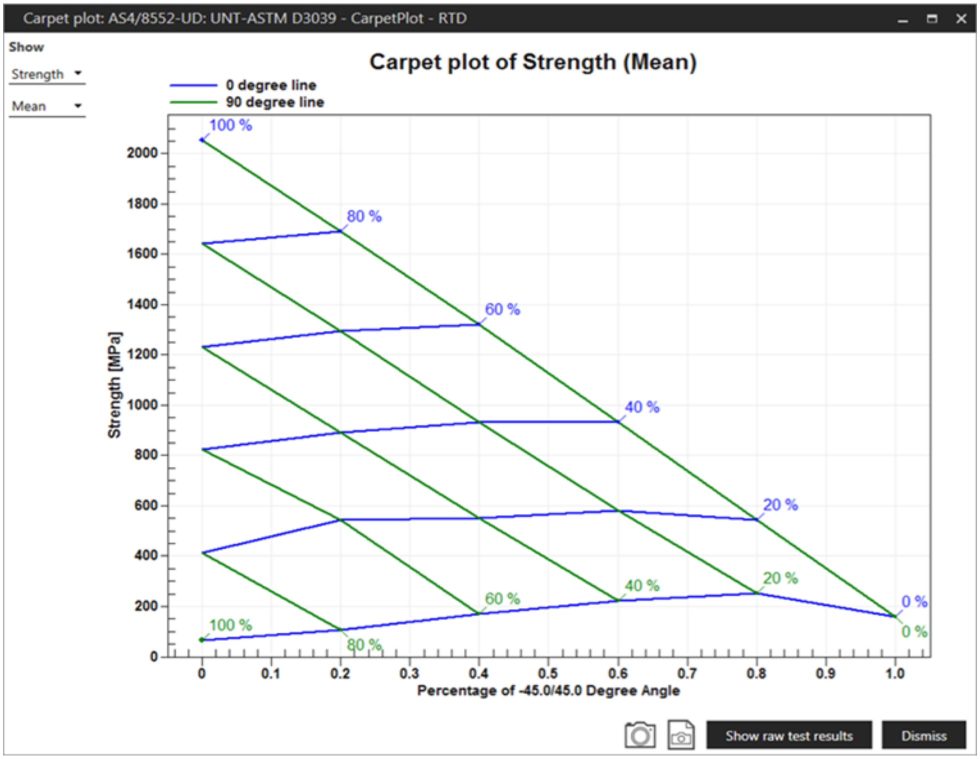


Figure 2-3 Example of a carpet plot.

To create this kind of plot in Digimat-VA, a special type of layup must be created: carpet plot layup (see Figure 2-4). Instead of specifying an exact stacking sequence, only three ply orientations need to be specified (called first, intermediate and last). In addition, the way in which proportion of each orientation should be varied is specified by means of the layup proportion increment parameter.

Based on that information, Digimat-VA will automatically generate the corresponding set of classical layups (as listed in the right part of the carpet plot layup creation window as shown in Figure 2-4). The following steps of analysis setup and run are strictly identical for classical or carpet plot layups.

For specific post-processing of carpet plots, see Carpet Plot.

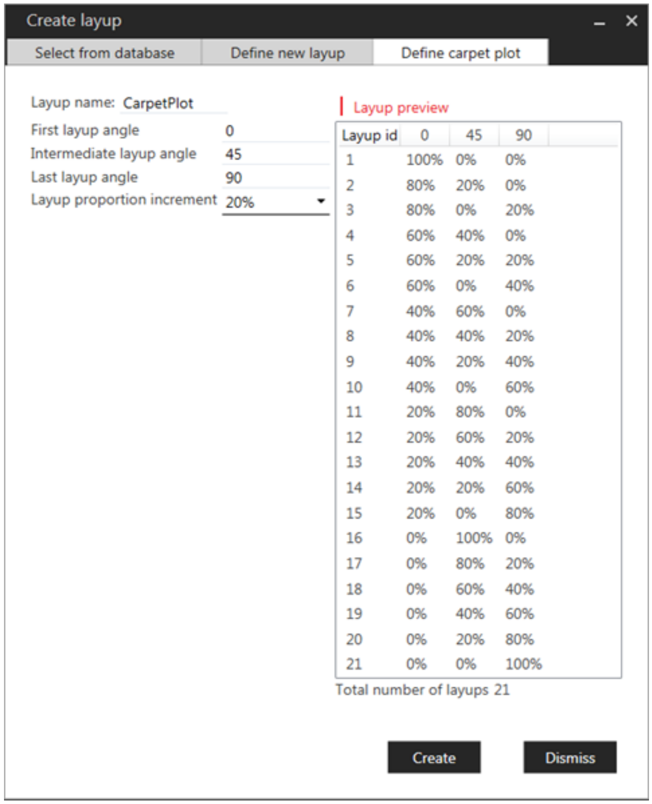


Figure 2-4 Carpet plot layup definition.

Standard Test Definition

Several types of standard tests representative for experimental composite characterization methods (Allowables and Composite Characterization) are available in Digimat-VA:

- Unnotched (tension or compression)
- Open hole (tension or compression)
- Filled hole (tension or compression)
- Double shear bearing (single or double fastener)
- Single shear bearing (1 or 2-piece; for 2-piece, single or double fastener, tension or compression, stabilized or not)
- In-plane shear (V-Notched or ± 45 tensile)
- Bending (3 or 4 point bending)
- Impact (Drop weight impact, Tension after impact, Compression after impact)
- Inter-laminar (Short-beam strength, Curved-beam strength, Double cantilever beam, Calibrated end loaded split, End notched flexure)

For each test type, as many norms as necessary can be defined. A norm definition contains the physical dimensions of the test coupon: length, width and hole diameter (for open hole coupons, filled hole coupons and bearing tests). The thickness is determined by the layup and by the cured ply thickness of the system. The norm can also contain information about fastener setup (for filled hole and bearing tests), loading plate setup and measurement strategy (for bearing tests).

Once a test is selected and its norm is specified, a test will be further identified via its acronym and the norm name. While the test acronym is straightforward for unnotched (UNT/UNC), open and filled hole (OHT/OHT and FHT/FHC), bending (3PB/4PB), drop weight impact (DWI), tension/compression after impact (TAI/CAI), short-beam strength (SBS), curved-beam strength (CBS), double cantilever beam (DCB), calibrated end loaded split (C-ELS) and end notched flexure (ENF); for bearing test it is worth detailing:

- Double shear bearing/single-fastener/tension: DSBSFT
- Double shear bearing/double-fastener/tension: DSBDFT
- Single shear bearing/two-piece/single-fastener/tension/unstabilized: SSB2PSFT
- Single shear bearing/two-piece/single-fastener/tension/stabilized: SSB2PSFTS
- Single shear bearing/two-piece/single-fastener/compression: SSB2PSFC
- Single shear bearing/two-piece/double-fastener/tension/unstabilized: SSB2PDFT
- Single shear bearing/two-piece/double-fastener/tension/stabilized: SSB2PDFTS
- Single shear bearing/two-piece/double-fastener/compression: SSB2PDFC
- Single shear bearing/one-piece: SSB1P

The standard test definition window is illustrated in Figure 2-5.

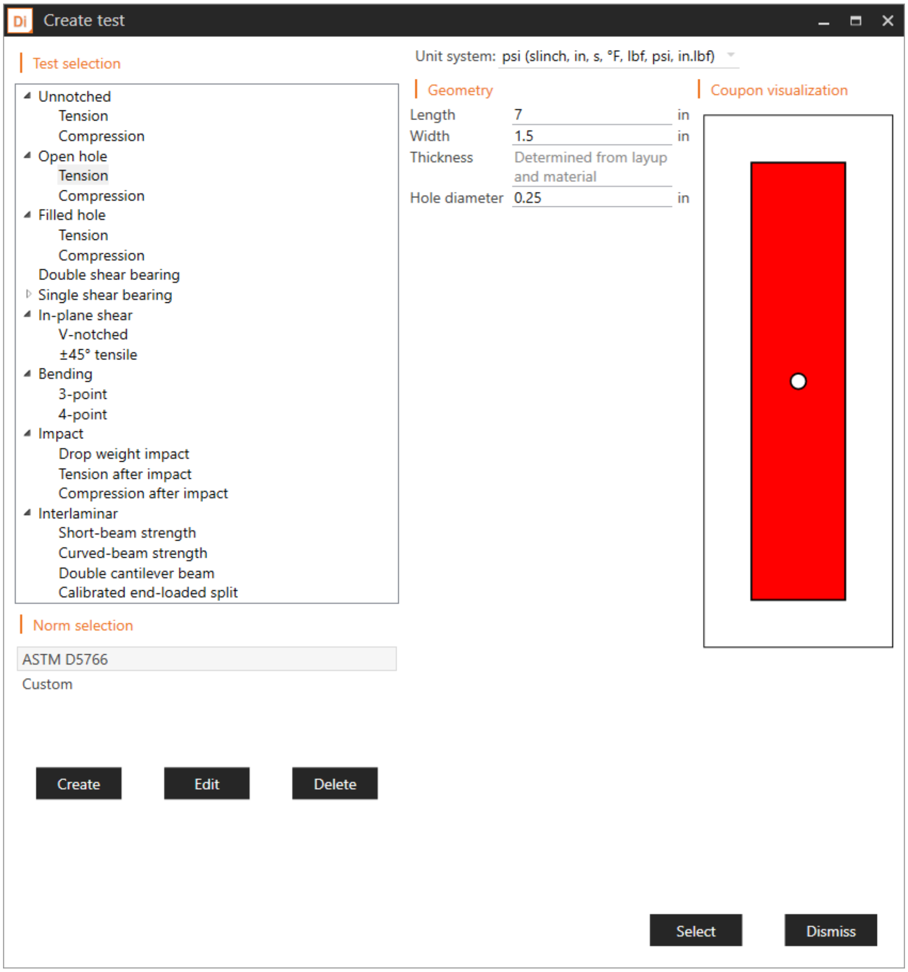


Figure 2-5 New standard test definition.

Unnotched Test

For unnotched test (tension and compression), an extra parameter, not purely geometrical, is available: the free length (see Figure 2-6).

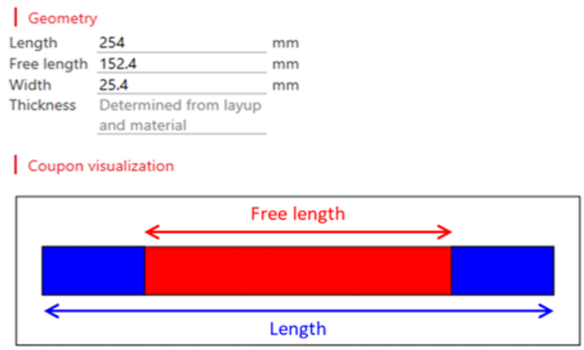


Figure 2-6 Meaning of the free length parameter for unnotched coupons.

The goal of that parameter is to define a region at the center of the coupon where the failure will be forced to occur (to avoid having failure driven by local stress concentrations arising at the boundary conditions). This is achieved by simply affecting a material model without damage outside of the center area. Physically, this free length reprensents the region of the coupon which is not held within the grips of the testing machine.

Bearing Test

Test Definition

To set up the test geometry, there are three or four tabs available in Digimat-VA depending on the type of bearing test. The parametrization for each of the tabs is explained below:

- Coupon
    - Length: Length of the coupon
    - Width: Width of the coupon
    - Hole diameter: Diameter of the hole (which is assumed to be the same for both holes in the case of ASTM D7248
    - Edge distance: Distance between the center of the hole (hole closest to the coupon edge) and the edge of the coupon
- Fastener
    - Fastener type: Protruding
        - Head and nut diameter: The head and nut are assumed to have the same diameter

  - Head and nut thickness: The head and nut are assumed to have the same thickness
  - Double shear bearing tests only support protruding fastener
- Fastener type: Countersunk/Countersunk (flat-edge)
  - Head diameter: Maximum diameter of the head of the countersunk fastener
  - Hole external diameter (only countersunk flat-edge): Diameter of the hole at the surface of the laminate
  - Countersink depth: May be computed from the head diameter, pin diameter and the countersunk angle
  - Head flat edge height (only countersunk flat-edge): Thickness of the countersunk fastener head which has a constant diameter
  - Nut diameter: Diameter of the nut
  - Nut thickness: Thickness of the nut
  - Countersunk fasteners are assumed to be flush with the laminate surface
- Pin diameter: Diameter of the shaft of the fastener
- Torque: Fastener tightening torque
- Young modulus (Fastener): Young's Modulus of the fastener material
- Poisson's ratio (Fastener): Poisson's ratio of the fastener material
- In the case of double fasteners, both fasteners are assumed to have the same type, geometry and material

■ Loading plate
- The loading plate is only available for double shear bearing tests and one-piece single shear bearing tests
- Length: Total length of the loading plate
- Width: Width of the loading plate
- Thickness (only double shear bearing): Thickness of the loading plates
- Minimum thickness (only single shear bearing - one piece): Minimum thickness is adjusted based on the thickness of the laminate to prevent eccentricity
- Length of thicker part (only single shear bearing - one piece): Length of the thicker part which is positioned in the clamps of the tensile testing machine
- Maximum thickness (only single shear bearing - one piece): Maximum thickness is adjusted based on the thickness of the laminate to prevent eccentricity
- Young modulus & Poisson's ratio: Elastic properties of the loading plate material (identical for both loading plates in the case of double shear bearing)

■ Miscellaneous

- This tab is used to define the position and type of strain gauges used to measure the elongation of the hole (used to compute the bearing strains)
- ASTM D5961 supports both Face and Edge strain gauge locations, but ASTM D7248 only supports Edge strain gauge locations in accordance with the respective norms
- In accordance with the norms, the offset strain to extract the bearing strengths may be defined and has a default value of 2%
- Illustration of the definitions for single and double fastener strain gauge locations are provided in Figure 2-7 and Figure 2-8.

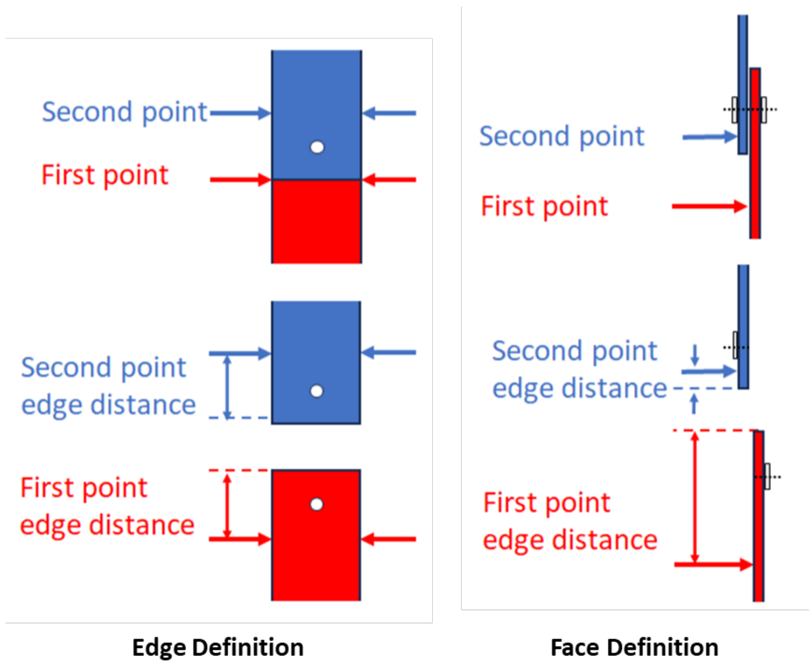


Figure 2-7 Bearing strain gauge definition for single fastener tests.

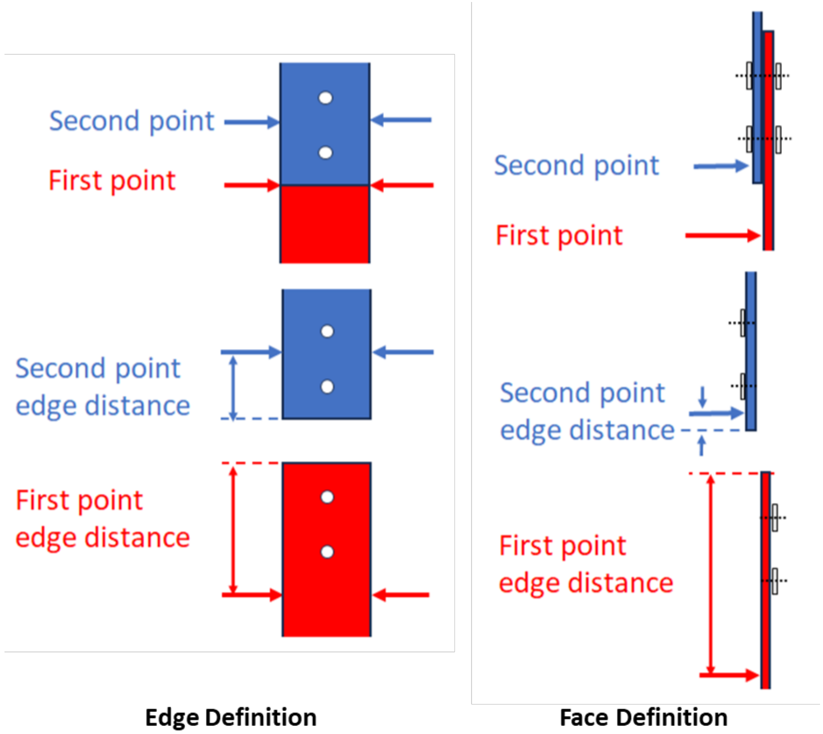


Figure 2-8 Bearing strain gauge definition for double fastener tests.

Bearing simulations with the internal solver can pose some convergence difficulties. This is due to damage occurring at the point of load introduction in the laminate (bearing failure in the hole). To overcome these issues, it is advised to switch from an instantaneous damage evolution to a linear damage evolution (see Extra Model Parameters Including Delamination).

Double Shear Bearing

The Double Shear Bearing test may be set up in accordance with either the norm ASTM D5961 (Single fastener) or ASTM D7248 (Double fastener). Only tension tests are available for this configuration. Each norm follows its respective method for results extraction and post-processing.

Single Shear Bearing

The Single Shear Bearing test may be set up in accordance with either the norm ASTM D5961 or ASTM D7248. The single shear bearing tests are broadly categorized into two-piece and one-piece fastened joints where two-piece consists of two identical laminates with a fastener connection and one-piece consists of a single laminate fastened to a loading plate. The two piece tests may be either joined with a single fastener or two fasteners and loaded in either tension or compression. The tension coupons may be either stabilized or un-stabilized where the stabilized coupons have an additional out-of-plane clamping system to reduce bending in the coupons. All compression tests use the stabilized configuration.

Bending Test

For bending tests (3 and 4 point), additional parameters need to be defined in accordance to the selected norm (see Figure 2-9):

- Region with damage: Option to choose if damage is applied on:
  - Full coupon length
  - Loading span (4PB only)
  - Support span
- Support radius
- Loading nose radius
- Support span: distance between supporting pins
- Support to load span ratio (Only for 4 point bending tests)

Depending on the norm selection, the appropriate post processing method chosen is displayed in the **Create test** window.

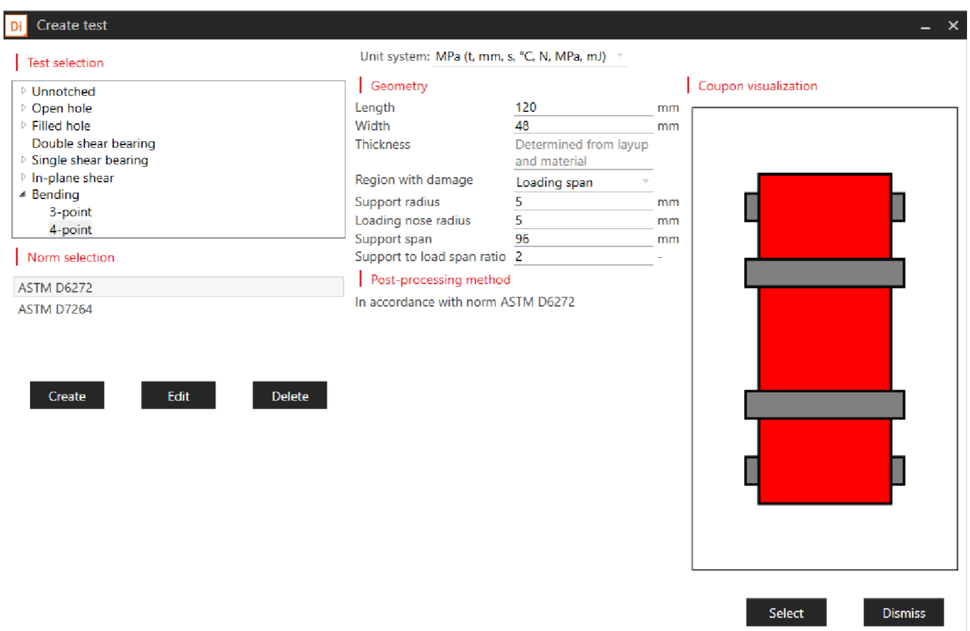


Figure 2-9 Test definition for bending.

Drop Weight Impact Test

The Test definition for impact requires the definition of geometries and testing conditions for multiple components. Each component has its individual tab. In the Coupon tab, the overall length and width of the laminate must be defined. Additionally, the dimensions of a central region must be provided to define the region where damage can occur. Outside of this region, appropriate material models will be assigned without damage definitions (see Figure 2-10).

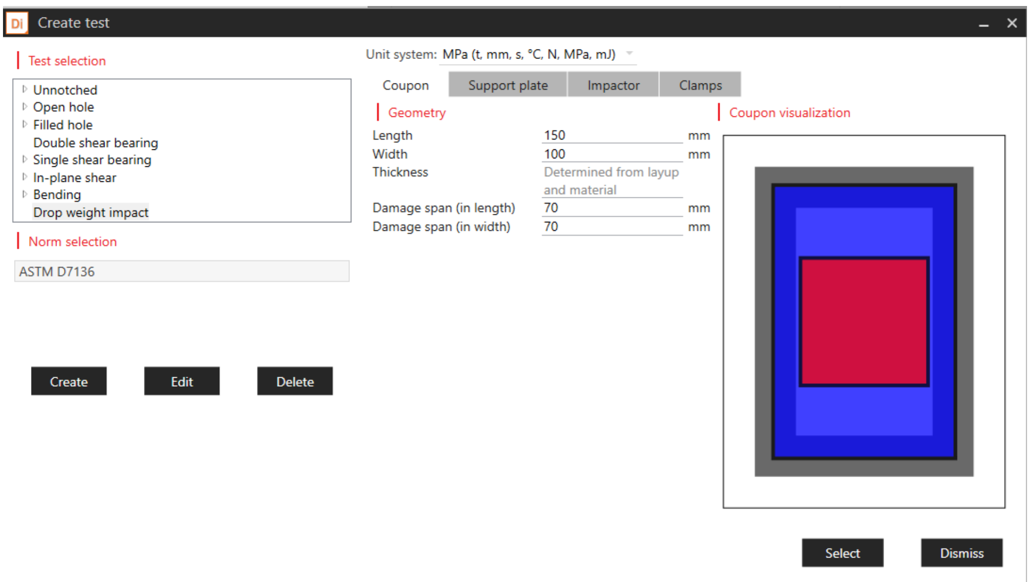


Figure 2-10 Test definition for Drop Weight Impact - Coupon.

In the Support Plate tab, the dimensions of the supporting plate and the cut-out window need to be defined. The supporting plate must be larger than the laminate and the cut-out window must be smaller than the laminate. See Figure 2-11.

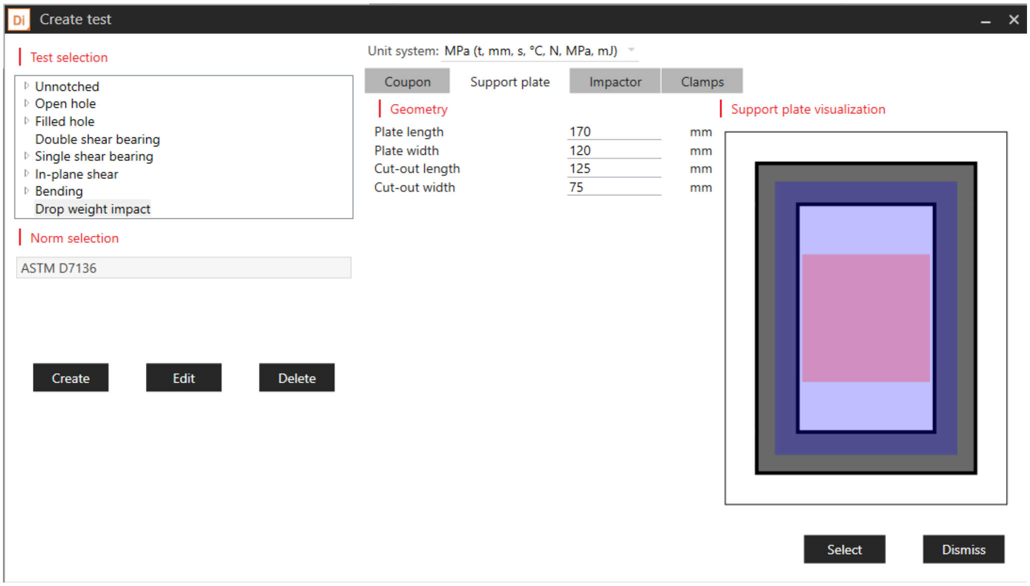


Figure 2-11 Test definition for Drop Weight Impact - Support Plate.

In the **Impactor** tab, the radius of the spherical impactor has to be defined. Additionally, the impactor velocity (at the point of impact) and the mass of the impactor needs to be defined. The impact energy will be computed automatically from these two parameters. See Figure 2-12.

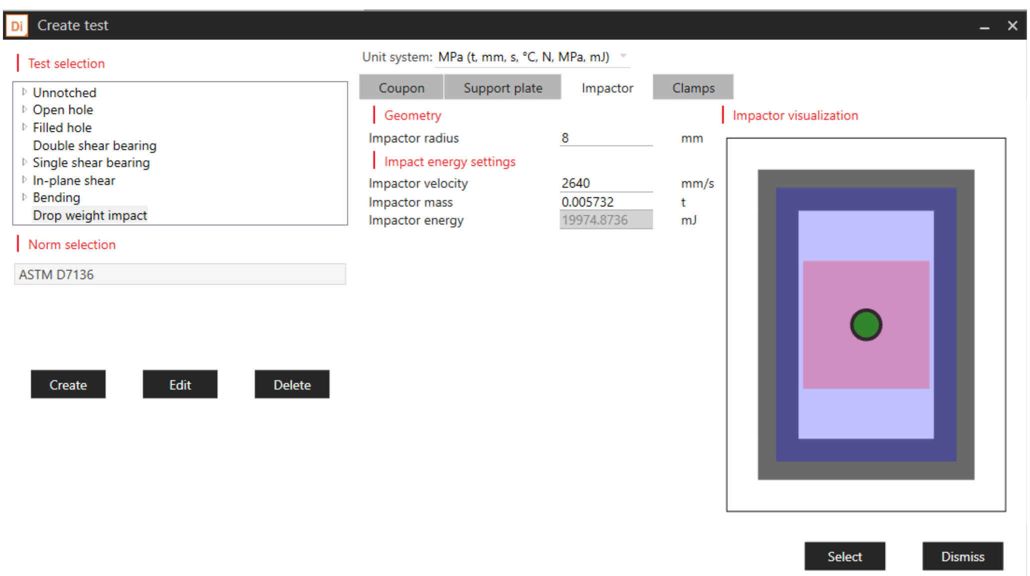


Figure 2-12 Test definition for Drop Weight Impact - Impactor.

In the Clamps tab, it is possible to choose between spherical or cylindrical clamps. The clamp radius and position relative to the support plate cut-out window will also need to be defined. It is also possible to define the clamping force which is applied by the clamps on the laminate (see Figure 2-13).

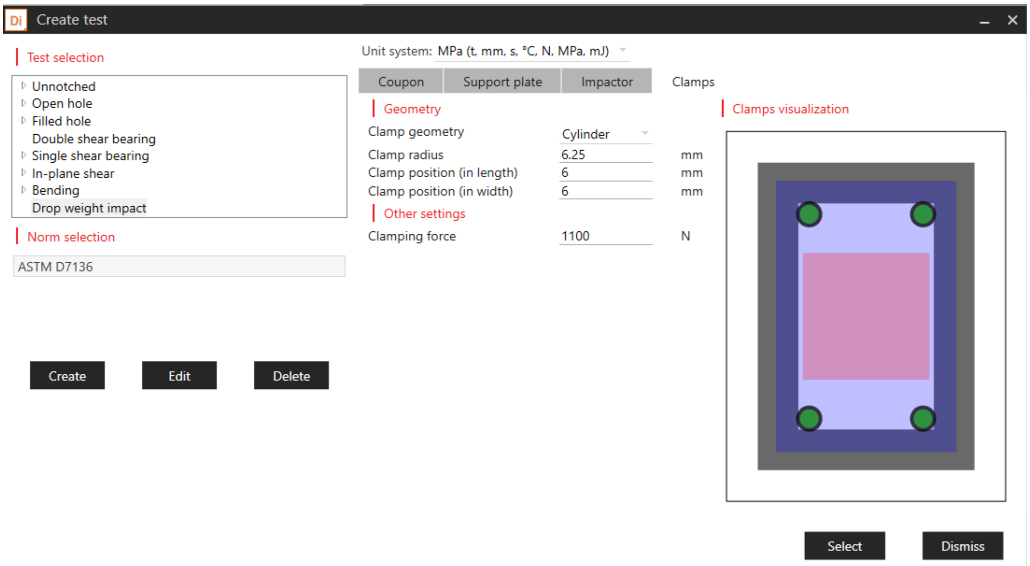


Figure 2-13 Test definition for Drop Weight Impact test - Clamps.

Tension After Impact Test

The tension after impact test is based on the drop weight impact test. The coupon geometry definition is identical. The only additional parameter concerns the way the tension load is applied on the coupon. A fifth tab is available to define this parameter (see Figure 2-14).

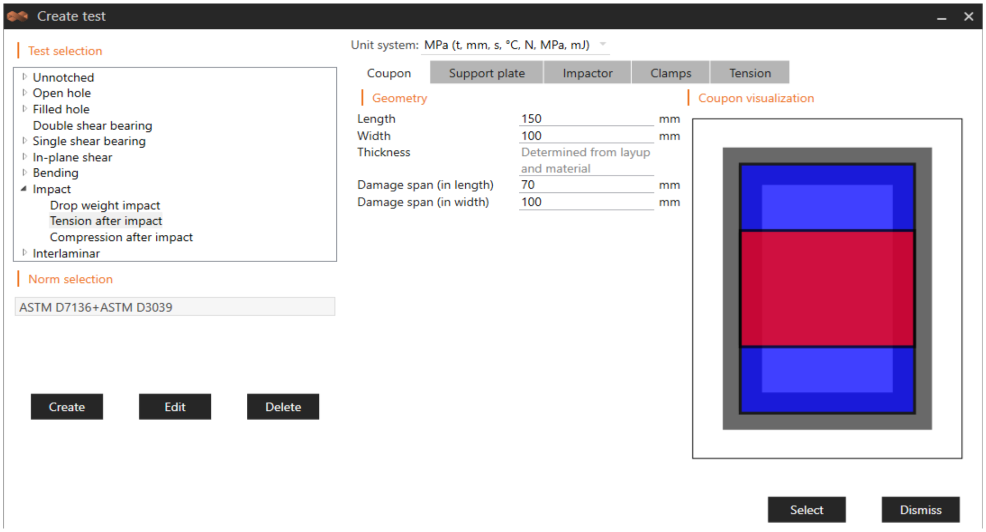


Figure 2-14 Test definition for Tension After Impact test - Tension.

The following parameter needs to be defined:

- End fixation height: the height of the area in green in Figure 2-14, this is the area on which the tension boundary conditions will be applied.

Compression After Impact Test

The compression after impact test is based on the drop weight impact test. The coupon geometry definition is identical. The only additional parameters concern; the way the compressive load is applied on the coupon. A fifth tab is available to define those parameters (see Figure 2-15).

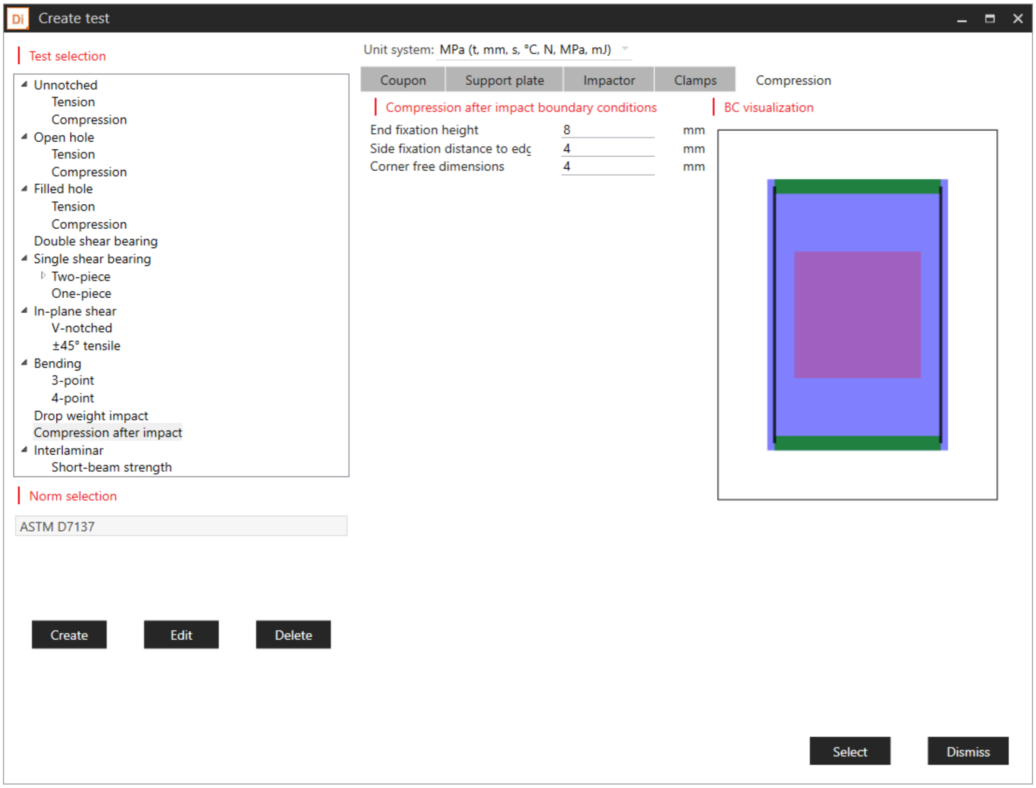


Figure 2-15 Test definition for Compression After Impact test - Compression.

The following parameters needs to be defined:

- End fixation height: the height of the area in green in Figure 2-15, this is the area on which the compression boundary conditions will be applied.
- Side fixation distance to edge: the distance between the edge of the coupon and the two black vertical lines in Figure 2-15, where a displacement of 0 will be applied in the out-of-plane direction during the compression step.
- Corner free dimensions: in each corner of the coupon, there is a small square area where no BC is applied. This parameter is the dimension of that square area.

Interlaminar Strength

Short-beam Strength Test

The Short-beam strength test is based on the 3-point bending test. The geometry of the coupon and the positions of the supports and indentor are changed so that the failure is governed by the delamination, therefore allowing to evaluate interface properties.

The main differences between Short-Beam Strength Test and 3-point bending test are:

1. The span is defined by mean of the span-to-thickness ratio instead of the direct input of an absolute span value.
2. There is no region without damage.

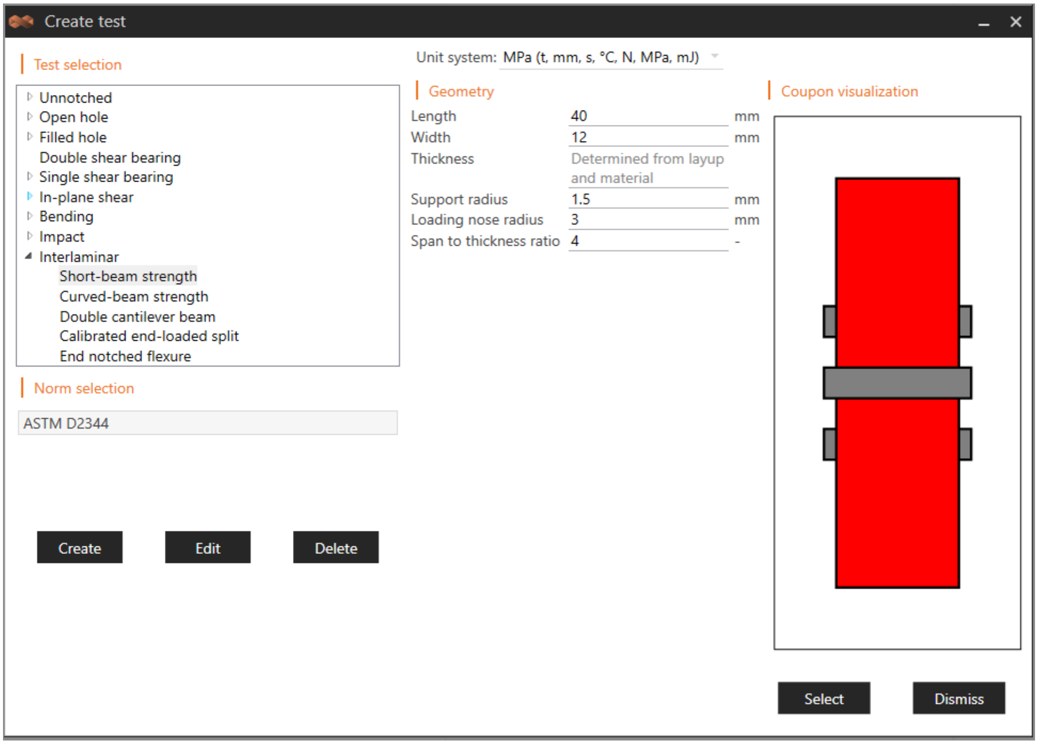


Figure 2-16 Test definition for Short-beam Strength test.

Curved-beam Strength Test

The curved-beam strength test is based on the 4-point bending test and is used to identify the interlaminar strength of a stacking sequence. The definition of the geometry for this test has been modified to suit the needs of the ASTM D6415 testing standard where the required geometry inputs are shown in Figure 2-17. Unlike in the case of the 4 point beam test, there is only a damageable region and delamination is present between all plies irrespective of the stacking sequence.

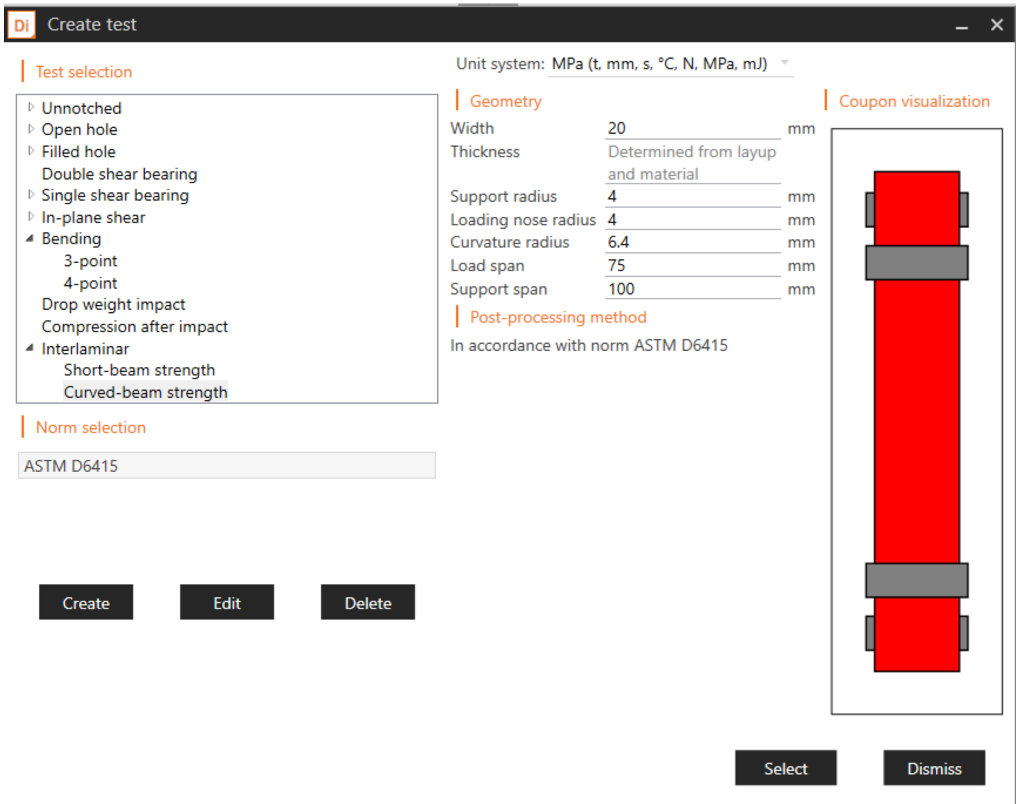


Figure 2-17 Test definition for Curved-beam Strength test.

Double Cantilever Beam Test

The purpose of the test is to measure the inter-layer Mode I fracture toughness. This test follows the testing standard ASTM D 5528-01. The specimen consists of a UD composite plate which has been manufactured with a crack of specific dimensions at the mid-plane of the stack. Attachments are fixed to the crack end position of the specimen on the top and bottom faces. These attachments are pulled apart in a tensile testing apparatus to grow the crack. The forces measured at the loading points, the opening displacement and the evolution of crack propagation are used to compute the fracture toughness.

There are two variants of this test as shown in the image below. (a) Piano hinge attachment, where the load introduction position is considered to be located at the middle of the pin of the hinge. (b) Loading Block attachment, where the load introduction position is considered to be the middle of the pin in the loading block. These load introduction positions are important to the calculation of the fracture toughness and may vary based on the dimensions of the hinge or loading block. The definition of the required specimen dimensions are show in Figure 2-18.

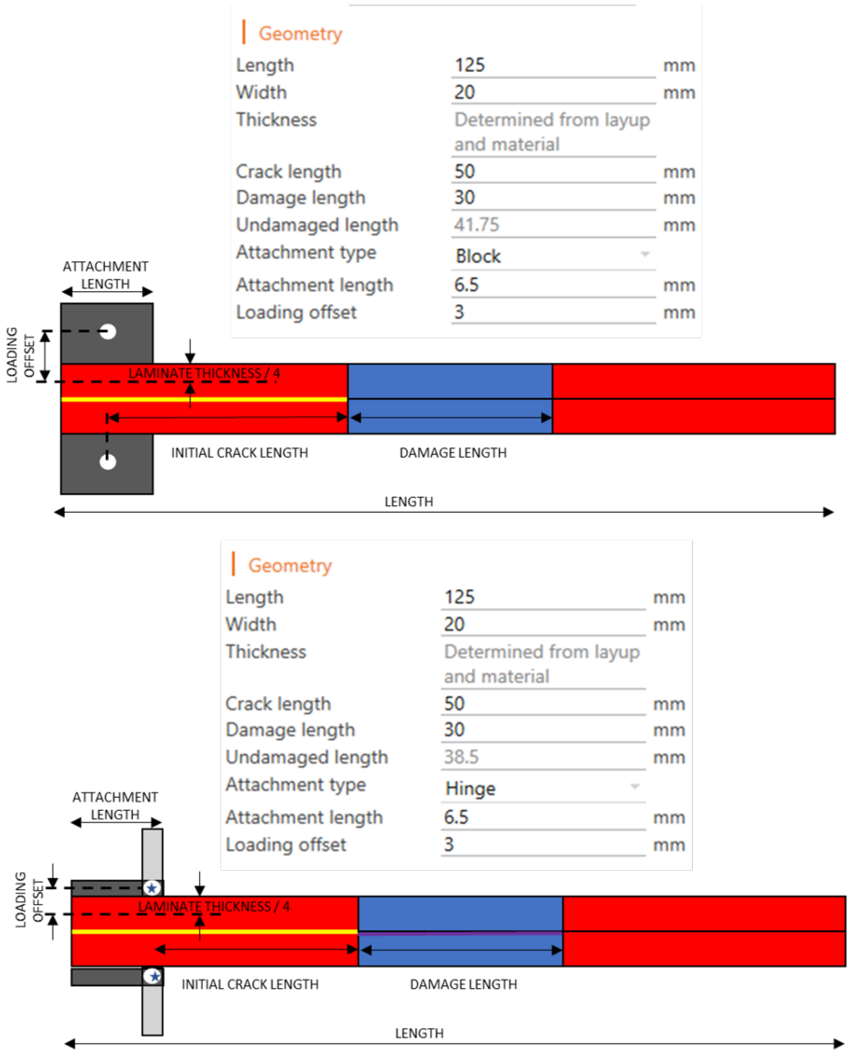


Figure 2-18 Test definition for Double Cantilever Beam Test.

Calibrated End Loaded Split Test

The purpose of the test is to measure the inter-layer Mode II fracture toughness. This test follows the testing standard ISO 15114. The specimen consists of a UD composite plate which has been manufactured with a crack of specific dimensions at the mid-plane of the stack. An attachment is fixed to the crack end position of the specimen on the bottom face. The laminate is loaded vertically at the attachment to facilitate an interface shear failure at the crack tip. The forces measured at the loading point, the vertical displacement and the evolution of crack propagation are used to compute the fracture toughness.

An illustration of the test and the geometry parameters can be seen in Figure 2-19.

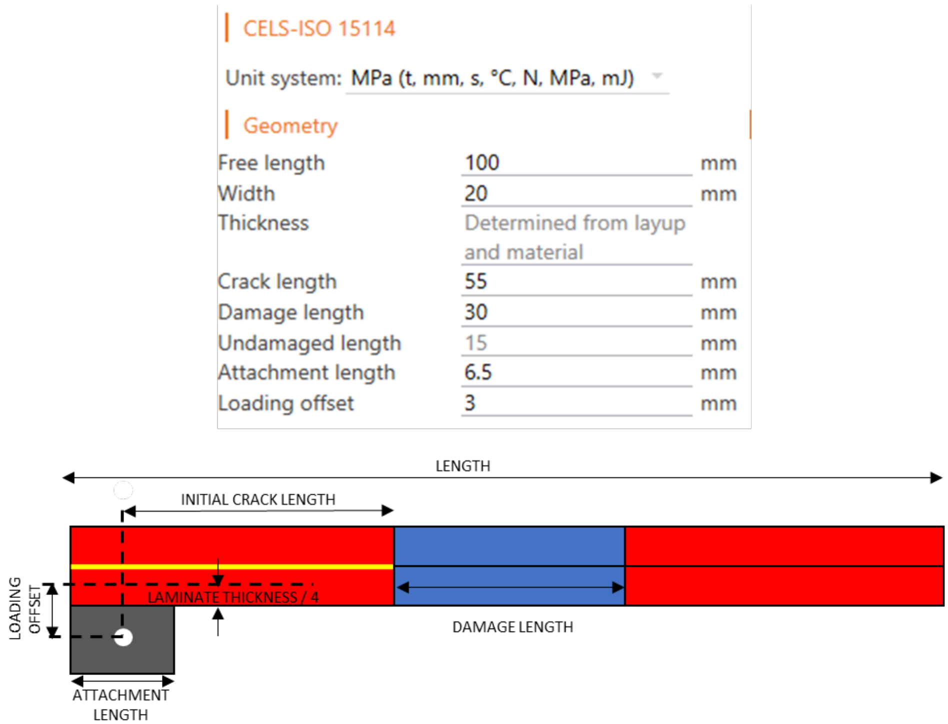


Figure 2-19 Test definition for Calibrated End Loaded Split Test.

End Notched Flexure Test

The End Notched Flexure Test is based on 3-points bending test but with a prepared crack in one side of the specimen, positioned at the mid-plane of the stack. The purpose of this test is to measure the inter-layer Mode II fracture toughness. A test standard as ASTM D7905 is followed during the implementation.

An illustration of the test and the geometry parameters can be seen in Figure 2-20.

Geometry

Length	130	mm
Width	20	mm
Thickness	Determined from layup and material	
Support radius	5	mm
Loading nose radius	5	mm
Support span	100	mm
Crack length	30	mm
Crack propagation length	25	mm

Coupon visualization

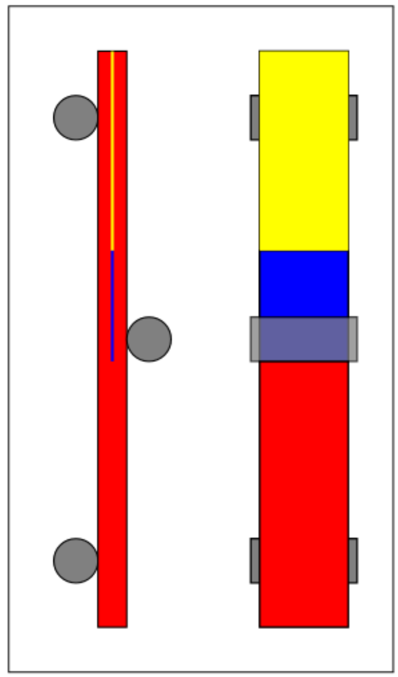


Figure 2-20 Test definition for End Notched Flexure Test

According to the standard, Compliance Calibration Method (CC) is used in the post-processing for computing Mode II fracture toughness. Using this, the compliance evolution when crack length increasing will be recorded and used for computing Mode II fracture toughness with the max force value before dropping in Force - Displacement curve.

Remark: This test assumes that the specimen has a high bending stiffness, hence a unidirectional stack with large thickness should be used in the test for a good prediction. Although in VA, the user is allowed to define any stack and any test configuration they wants, for the sake of validating their test configuration.

Environment Conditions

Environment conditions have the following data as shown in Figure 2-21.

- Name
- Temperature
- Relative humidity

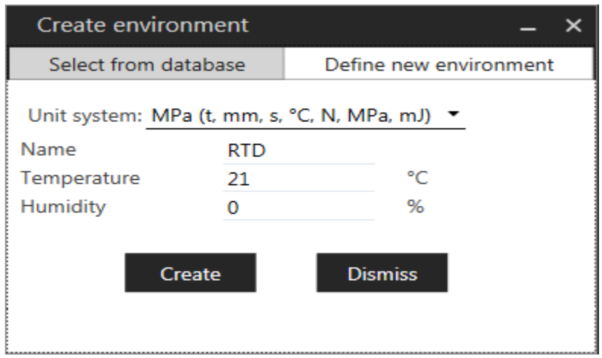


Figure 2-21 New environment conditions definition.

Test Matrix Edition

Once all items (i.e., materials, layups, tests and environments) have been added to the test matrix definition, virtual tests are created for each and every combination of the test matrix items.

A visual representation of this set of virtual test (the test matrix) can be accessed by clicking on the matrix icon in the lower left corner (see Figure 2-22). The full set of virtual tests is displayed in tables, grouped by material (one table per material), then by environment conditions (one column per environment), then by combination of test and layup (one table row for each).

This kind of display will be used throughout Digimat-VA. In this window, each individual virtual tests can be enabled or disabled. All disabled test will be ignored in the following steps of the workflow.

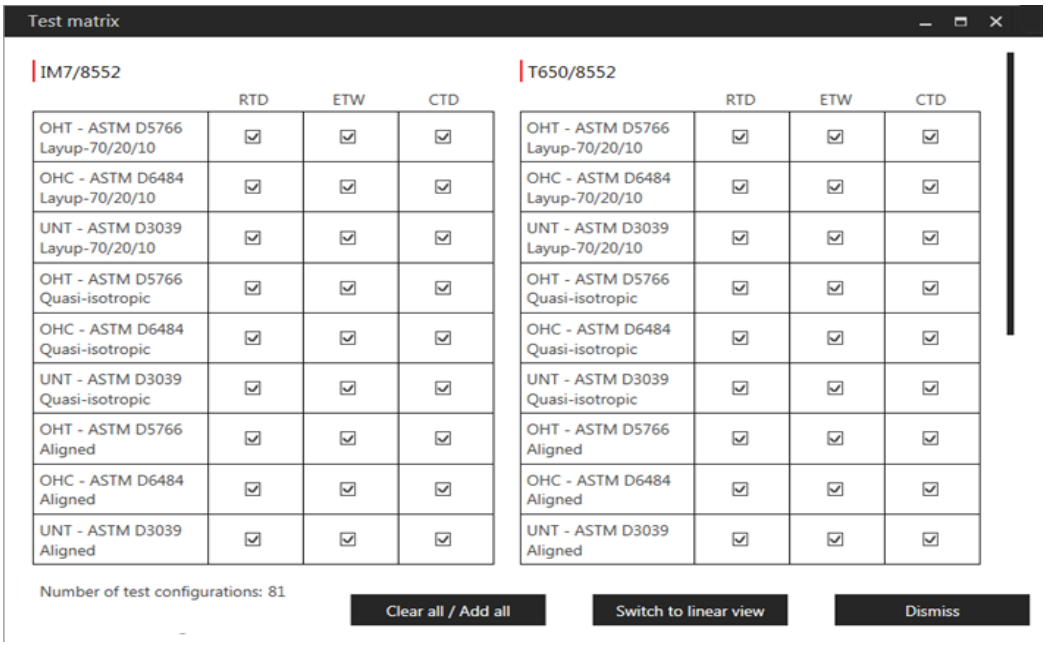


Figure 2-22 Test matrix edition.

Variability definition

One of the key features of Digimat-VA is the capability to model the variability of material, process and testing related parameters. This inherent variability of composite behavior can be characterized, typically by computing mean values, A and B-basis values from a given sample of the population. For details about the computation of the statistical values, please refer to Computation of Allowables

In Digimat-VA, the user can choose between a simple approach (no variability), a stochastic approach (Standard scenario according to MIL HBK) or two types of deterministic approach (parametric study or defect study) as shown in Figure 2-23. No variability allows to run one single simulation per configuration, for instance predicting laminate mean response based on mean lamina data. It is recommended to start a Digimat-VA campaign with a simple approach, in order to check the validity of the defined simulations (applied strain, unit system...).

When choosing the **Standard scenario according to MIL HBK** option, you can define the number of batches, panels and specimens you wish to test. In the Digimat-VA framework, material properties variability is associated to batch definition. Process parameters such as fiber volume fraction are associated to panel definition.

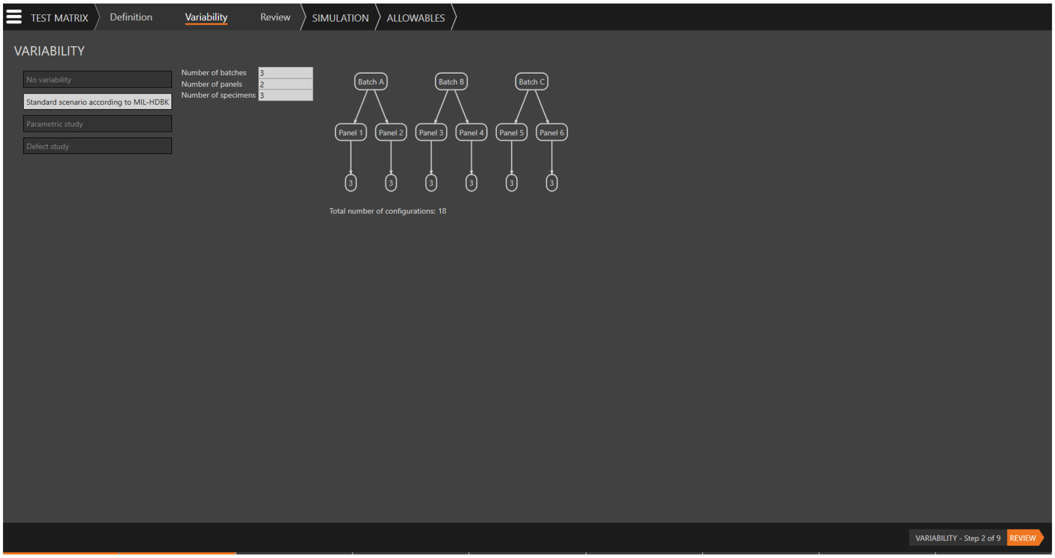


Figure 2-23 Variability definition in Digimat-VA.

Testing related parameters, such as fiber misalignment (inherent to misalignment of specimen in the fixtures) are associated to specimen definition. This means that if defining several batches, the user considers material properties will change from one batch to the other one, if defining several panels, fiber volume fraction will change from one panel to the other one, etc.

If the user defines, lets say three batches, two panels per batch and three specimens per panel, it means that in the simulation part, based on the variability model of the Digimat model,

- three draws of the matrix and fiber properties will be performed
- six draws of fiber volume fraction will be performed
- eighteen draws of fiber misalignment will be performed

These will lead to eighteen different definitions of Digimat models, which will be used in the eighteen simulations which are required by defining such a sampling of batch/panel/specimen.

When choosing the **Parametric study** option, the user opens a workflow in Digimat-VA which will further enable him during the Simulation step to define the parameters to include in the parametric study. Available parameters include material parameters such as constituent stiffness or strength and fiber volume fraction, layup parameters such as ply misalignment as well as some test parameters such as coupon length, width or hole diameter.

The workflow of the **Defect study** option is very similar to the **Parametric study** option. In the **Simulation** step, it will be possible to define different types of defect and to assign them individually to the different coupons in the test matrix (see FE Analysis for details).

3 Simulation

- General Description
- Material Model
- FE Analysis
- FEA Job Submission

General Description

The simulation step in Digimat-VA consists of three substeps: Digimat model, FEA analysis and Job submission. The two first substeps can require some user input, and allow to build the desired simulations, which require the definition of both a Digimat material model including the desired modeling approach and finite element models of the requested coupon tests. Once the simulations are ready, they can be run and monitored in the last substep.

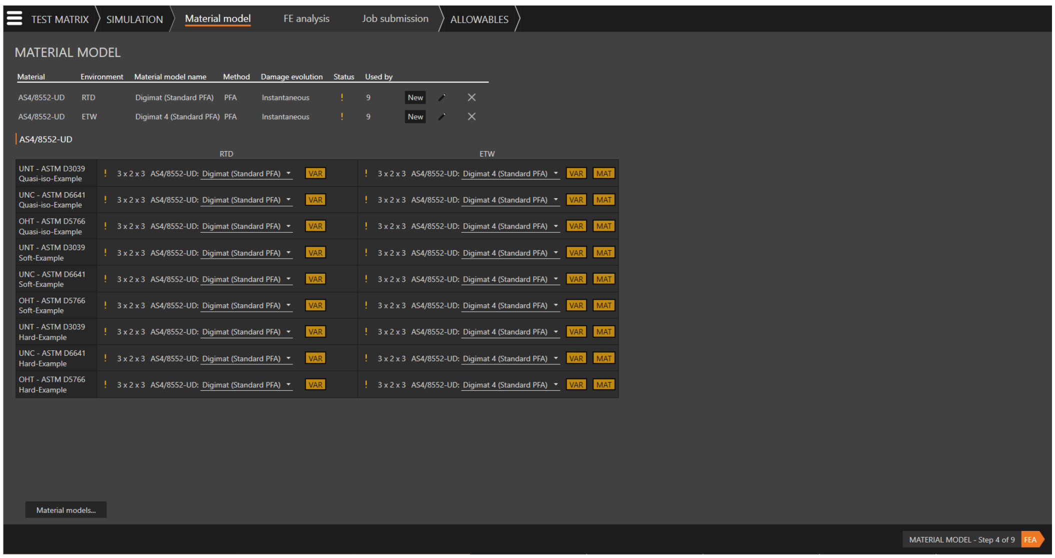


Figure 3-1 Digimat model screen.

Material Model

Digimat-VA currently offers 3 different types of material model:

- **Standard PFA (Standard Progressive Failure Analysis):** A progressive failure method is used. It relies on the 2D damage model of Matzenmiller (see Damage Models) for unidirectional materials and on the 2D multi-component damage model (see Damage Models) for woven materials. The FEA jobs are terminated once a significant load drop is detected on the force displacement curve.
- **Advanced PFA (Advanced Progressive Failure Analysis):** Another progressive failure method is used. It relies on the Camanho damage model (see Damage Models) and takes into account the manufacturing induced stresses as well as the so-called in-situ strengths.

Compared to the Standard PFA, it requires more input data and is more expensive from the computational point of view but it yields more accurate predictions. The FEA jobs are terminated once a significant load drop is detected on the force displacement curve. This modeling strategy is currently only offered for unidirectional materials and some of the coupon geometries.

- **FPF (First Ply Failure):** a Tsai-Hill 3D Transversely Isotropic failure indicator is used in each ply (see Failure Indicators). Note that in Digimat-VA for the Tsai-Hill 3D transversely isotropic failure indicator, the tensile and compressive strengths in axial and in-plane direction are NOT assumed to be identical. With this failure modeling strategy, the FEA jobs are terminated as soon as the first element reaches failure.

The material model screen (see Figure 3-1) allows to check that a suitable material model exists for each and every virtual test, and to select the material model to use if there is more than one suitable model available. All virtual tests for which there is no suitable material model available are clearly identified by an orange check mark and boxes indicating what type of data is missing: material model, variability model or both.

Similar information is summarized in the status column of the table at the top of this screen, where a review of all material models used for the test matrix is summarized. The table aims at managing material models in a global fashion across all types of tests, while working directly in the test matrix enables to control precisely per test configuration the material modeling choices. Clicking on one of the **Edit** icon in the Material model table or clicking on the icons in the test matrix will open the material calibration window for the corresponding configuration.

When a suitable material model exists, its name is displayed in the box. Right-clicking on one box will show a context menu allowing to select the material model to use (when more than one suitable model exist).

When CLT computations are activated, each box has a **CLT** button that opens up a new window showing the detailed CLT results (stiffness, CTE, CME and first ply failure strength) for the selected layup and material (see Figure 3-2).

The **Disable incomplete configurations** button automatically disables all virtual tests for which there is no suitable material model available.

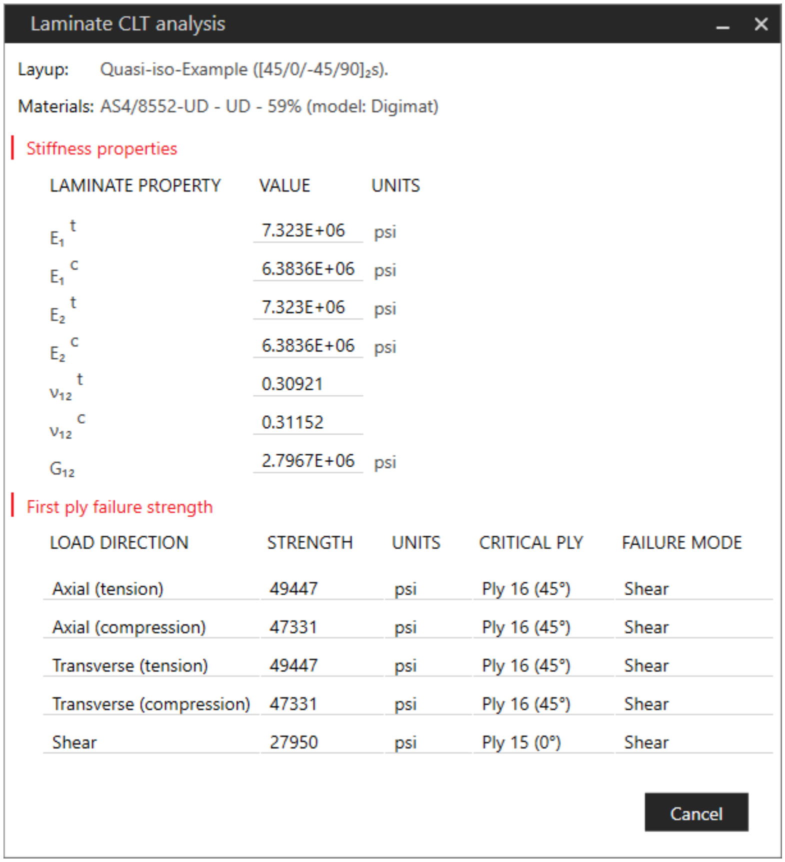


Figure 3-2 CLT results window.

Digimat Material Model Calibration

Virtual tests require preliminary Digimat material model calibration. Such calibration mainly consists in reverse engineering the material system constituent properties constituting a Digimat material model based on experimental data at lamina level. Indeed Digimat material models involve a combination of mean-field homogenization (see Mean-field Homogenization (MFH)) and a failure modeling strategy (either progressive failure, see Progressive Failure Model, or first ply failure).

Note that this calibration step is necessary even if only CLT computations are to be performed. The reason is that it allows to specify experimental ply properties at various fiber volume fraction. The calibrated Digimat model is then used to compute the ply properties at the nominal volume fraction, and those properties will then be used as input in the CLT computations.

From a user perspective, the material model calibration follows 4 successive steps.

- Select the failure modeling strategy by clicking on **Create new material model** (see Figure 3-3): either Standard PFA, Advanced PFA or FPF.
- Input the experimental data needed for calibration purposes in the **Experimental data** tab (see Figure 3-3).
- Click on **Calibrate material model** to proceed with the reverse engineering of the constituent properties which, upon successful completion, will switch from the **Experimental data** tab to the **Digimat material model** tab (see Figure 3-4)

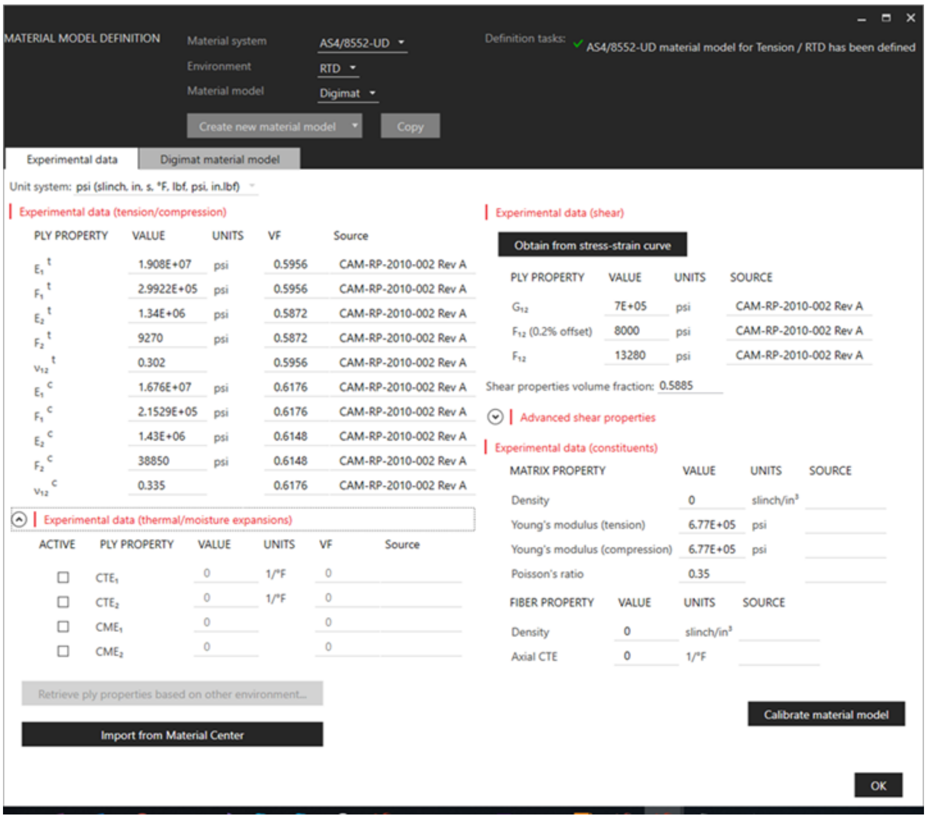


Figure 3-3 Input of the experimental data needed for the calibration of the Digimat material model.

- Check the calibrated constituent properties in the **Digimat material model** tab (see Figure 3-4) and override their values manually (if needed). This tab also makes it possible to visualize the resulting stress-strain behavior for different fiber volume fractions and different loading conditions.

A calibrated Digimat material model reproduces the experimental data at their respective volume fraction. Few exceptions produce a model not accommodating a limited subset of the input experimental data.

These exceptions include cases when the difference between several data is not considered significant, the consistency between several data is questionable or a Digimat material model cannot reproduce the full experimental data set.

- Woven materials exhibit small transverse Poisson's ratios ν_{12} by comparison to UD. Nevertheless not any value can be reproduced by a Digimat material model, either because of a questionable input value or a material model limitation. In such a case, the model is assigned with generic phase properties and the corresponding composite Poisson's ratio is displayed to the user.

The experimental data required for material model calibration mainly consist of lamina properties of the material system. As they most often depend upon the fiber volume fraction (VF), the latter needs to be specified for each lamina property given as input.

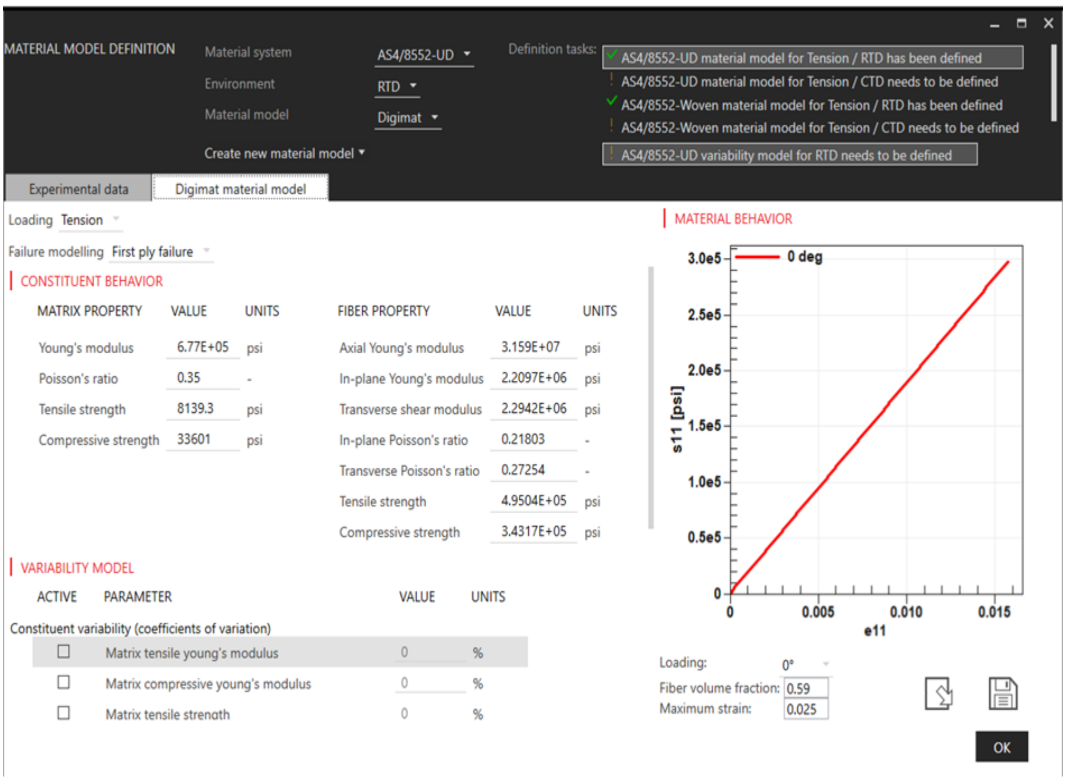


Figure 3-4 Parameters of the Digimat material model.

Lamina properties are typically collected during material characterization campaigns following dedicated standards (e.g., see Figure 3-5 from NCAMP Hexcel 8552 IM7 Unidirectional Material Property Data Report, p. 33):

Prepreg Material:	Hexcel Corporation - Hexcel 8552 IM7 Unidirectional NMS 128/2 Material Specification			**Hexcel 8552 IM7** **Unidirectional Tape** **Lamina Properties Summary**
Fiber	IM7 unidirectional	**Resin**	Hexcel 8552	
Tg(dry)	406.43 °F	**Tg(wet)**	321.41 °F	
PROCESSING:	NPS 81228 "M" Cure Cycle		**Tg METHOD** DMA (SRM 18-94)	

	Lot 1	Lot 2	Lot 3		
Date of fiber manufacture	01/26/2007	12/25/2006	02/05/2007	**Date of testing**	1/22/2008 - 3/4/10
Date of resin manufacture	02/28/2007	01/24/2007	03/01/2007	**Date of data submittal**	4/5/2010
Date of prepreg manufacture	02/28/2007	01/24/2007	03/01/2007		
Date of composite manufacture	9/2007 to 10/2007				

LAMINA MECHANICAL PROPERTY SUMMARY
Data reported as: Normalized & Measured
(Normalized by CPT= 0.0072 inch)

	CTD Mean		RTD Mean		ETD Mean		ETW Mean	
	Normalized	Measured	Normalized	Measured	Normalized	Measured	Normalized	Measured
F_1^{tu} (ksi)								
from LT	357.39	353.70	362.69	371.08	---	---	333.50	327.96
from UNT0	286.78	281.57	324.62	320.79	---	---	346.85	340.46
E_1^t (Msi) of LT	22.57	22.33	22.99	23.51	---	---	24.00	23.77
E (Msi) of UNT0	11.92	11.71	11.99	11.85	---	---	11.94	11.74
ν_{12}^t		0.270		0.316				0.393
F_2^{tu} (ksi)	---	9.60	---	9.29	---	---	---	3.49
E_2^t (Msi) of TT	---	1.46	---	1.30	---	---	---	0.81

Figure 3-5 Lamina mechanical property summary sample.

Tensile properties, e.g., measured according to ASTM D3039 (superscript t)

- Compressive properties, e.g., measured according to ASTM D6641 (superscript c)
- Shear properties, e.g., measured according to ASTM D3518

In Digimat-VA, these properties refer to the longitudinal/warp (resp. transverse/weft) directions when they are assigned with a subscript 1 (resp. 2). Most lamina properties consist in elastic moduli and strengths. Indeed, they refer to material behaviors assumed linear until failure, e.g., in tension. When the material behavior is significantly nonlinear, e.g., in shear, a modulus and a strength are obviously insufficient to fully characterize the material behavior. Hence shear test standards recommend to collect the following properties (see Figure 3-6). The data can either be inputed manually or extracted from the stress-strain curve.

The extraction of the shear ply properties from the stress-strain curve is done in a separate window (see Figure 3-7). The stress strain curve can be imported from the clipboard, from a file or directly copy paste in the table. The **Extract ply properties** button will compute the shear ply properties. The **Apply** button will apply the computed shear ply properties in the material model definition window.

Figure 3-6 Automated extraction of the shear ply properties from the shear stress-strain curve.

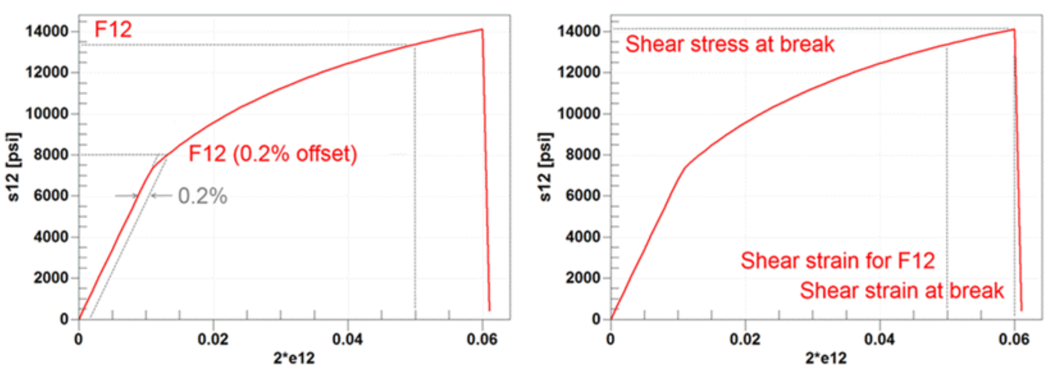


Figure 3-7 Shear properties sample, standard (left) and advanced (right).

The shear ply properties are described below.

- G_{12} denotes the chord shear modulus of elasticity. It is typically inferred from shear stress measurements at 0.2% and 0.6% strains.
- F_{12} (0.2% offset) denotes the 0.2% offset shear strength. Unlike its naming, it does not refer to a failure characteristic but more to a measure of nonlinearity. It is obtained by translating the shear chord modulus of elasticity line along the strain axis from the origin by 0.2% and determining the shear stress at the intersection of this line with the stress-strain curve.
- F_{12} denotes the (measured) maximum shear stress. It does not necessarily correspond to the actual shear strength, i.e., stress at break, but rather to the stress at a predefined strain, as usually recommended by shear test standards. Such recommendation may be interpreted as yielding a conservative strength measure, still reasonable taking into account the small variability of the shear stress to a shear strain variation due to the material nonlinearity.

To fully characterize the shear behavior, Digimat-VA involves modeling assumptions. These assumptions can be customized by editing the advanced shear properties.

- The shear strain for F_{12} reflects the predefined strain at which F_{12} has actually been measured. By default, it amounts to 5%, according to ASTM D3518.
- The shear strain at break corresponds to actual lamina failure. By default, it amounts to 6%. For larger strains, a dedicated damage evolution – instantaneous by default – is triggered as post-failure behavior.
- The shear stress at break corresponds to actual lamina failure. By default, it is computed based on F_{12} (0.2% offset) and F_{12}.

In order to calibrate a physical model of the shear behavior, F12 and the shear stress at break must belong to the range defined by the corresponding lower and upper bounds as illustrated in Figure 3-8.

In some cases, extra data need to be input in relation to thermal and/or moisture expansion. More particularly,

- If CLT computations have been activated, an extra section for experimental CTE and CME of the ply appears, as well as data fields for fiber and matrix densities and fiber axial CTE (see Figure 3-3). Those properties are optional, meaning that leaving them empty will not prevent calibration. It will simply disable the CLT computations for the missing property types (i.e. CTE, CME or both).

Note:	The fiber and matrix densities are required only for CME homogenization.

- When calibrating an Advanced PFA model, the experimental CTE of the ply, the curing temperature and the fiber axial CTE must always be provided as inputs. They can however all be set equal to 0 which will discard the manufacturing-induced stresses from the analyses.

In case the required experimental data has been stored in MaterialCenter it is also possible to directly import the data in the Experimental tab by selecting the appropriate file upon clicking on **Import from MaterialCenter**. Further details on the usage of the Digimat-VA interface to MaterialCenter are available in the dedicated section Interface to MaterialCenter

If allowables need to be generated for different environmental conditions, the lamina properties for different environmental conditions can be evaluated starting from a given set of lamina properties. In order to do so, one has to click on the Retrieve ply properties based on other conditions button. The new displayed window give access to additional parameters, like the glass transition temperature, that will drive the evaluation for the targeted environmental conditions. Some parameters are automatically suggested by Digimat, such as the resin shear strength from the Digimat model and the magnifications factors. These values are proposed as reference, user is left the opportunity to use alternative values coming from literature or his own experience.

The underlying model used in Digimat is based on the work of Chamis et al. (1978). Please note that this capability is only available when modeling unidirectional composites.

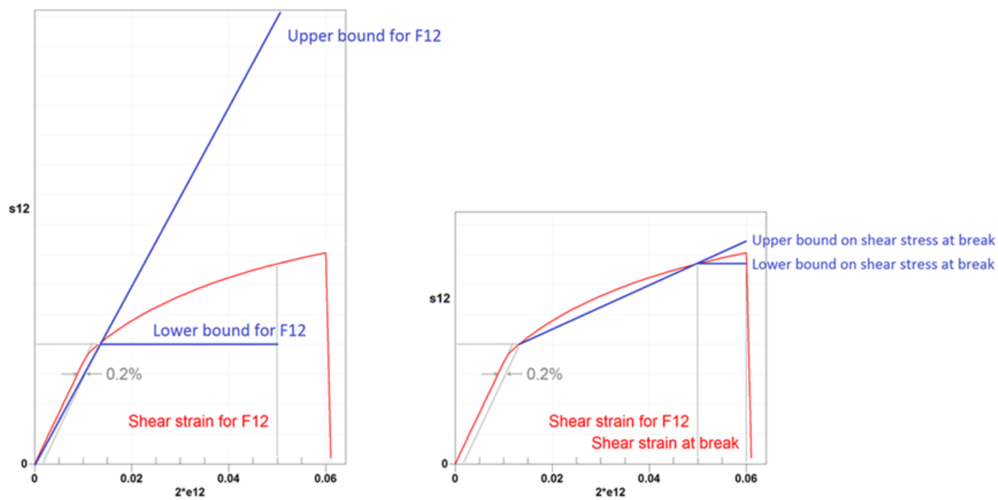


Figure 3-8 Conditions imposed on F_{12} and the shear stress at break.

Extra Model Parameters Including Delamination

Upon successful calibration of the material model, extra model parameters can directly be input in the **Digimat material model** tab. When using Standard PFA models, the extra model parameters consist of (see Figure 3-10):

- The type of damage evolution law: either instantaneous or with linear softening. In the latter case, the ratio of the strain at complete failure to the strain at damage initiation also needs to be input.

- The type of behavior at the interface between the plies: either without damage or with delamination capabilities. In the latter case, delamination is modeled with cohesive elements obeying the traction-separation behavior proposed by Turon et al. (2018) which itself relies on the energy-based criterion of Benzeggagh-Kenane for damage evolution (see Cohesive Material & Debonding). It requires defining the following extra parameters:
  - G_I = the fracture energy in normal mode only
  - G_{II} = the fracture energy in shear mode only
  - T_I = the interface strength in normal mode only
  - T_{II} = the interface strength in shear mode only
  - the Benzeggagh exponent
  - Friction coefficient (only available for DWI tests)
  - Critical damage for element deletion (only available for DWI tests)

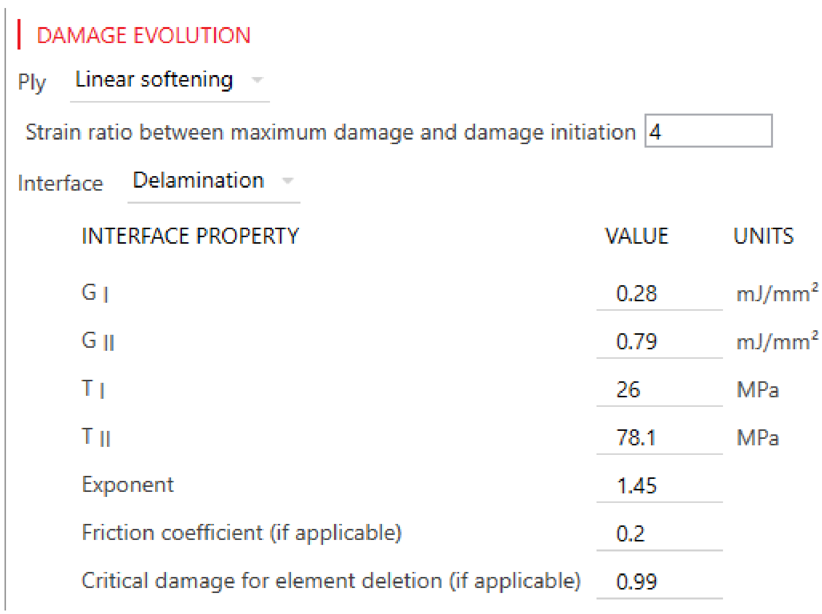
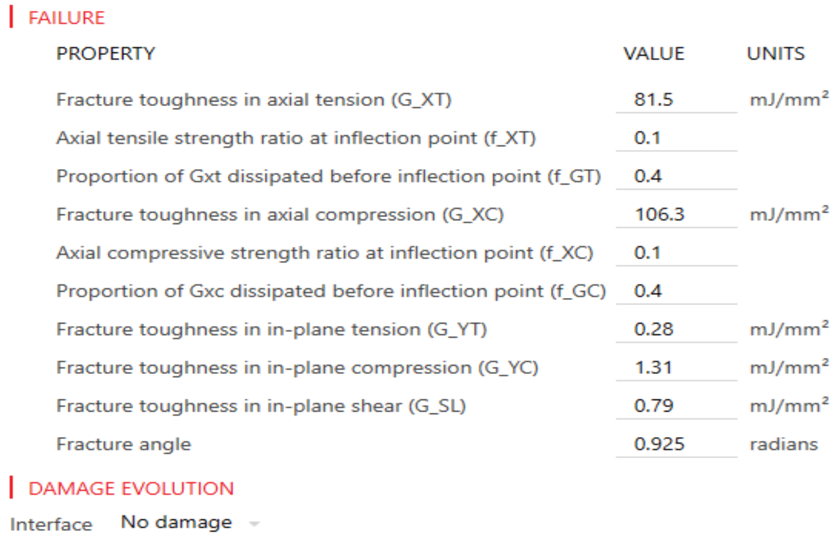


Figure 3-9 Extra model parameters which can be defined for the Standard PFA models.

When using Advanced PFA models, the extra model parameters consist of (see Figure 3-10):


FAILURE

PROPERTY	VALUE	UNITS
Fracture toughness in axial tension (G_XT)	81.5	mJ/mm²
Axial tensile strength ratio at inflection point (f_XT)	0.1	
Proportion of Gxt dissipated before inflection point (f_GT)	0.4	
Fracture toughness in axial compression (G_XC)	106.3	mJ/mm²
Axial compressive strength ratio at inflection point (f_XC)	0.1	
Proportion of Gxc dissipated before inflection point (f_GC)	0.4	
Fracture toughness in in-plane tension (G_YT)	0.28	mJ/mm²
Fracture toughness in in-plane compression (G_YC)	1.31	mJ/mm²
Fracture toughness in in-plane shear (G_SL)	0.79	mJ/mm²
Fracture angle	0.925	radians

DAMAGE EVOLUTION

Interface No damage

Figure 3-10 Extra model parameters which can be defined for the Advanced PFA models.

A total of 10 extra parameters are required for the Camanho damage model (see Progressive Failure Model):

It is recommended to obtain these parameters by an experimental testing campaign. However, if this is not possible, it is possible to compute sufficiently accurate parameters from more commonly available material data.

- Fracture toughness in longitudinal tension (G_XT):
  - Reverse engineer the value of the fracture toughness in axial tension (G_XT) by matching the open hole strength in tension of a quasi-isotropic layup. This could be set up as a project in Digimat VA by varying the G_XT value.
  - As a starting value, G_XT value for AS4/8552 in the Digimat VA database (81.5 mJ/mm^2) could be used.
  - The default values for the longitudinal tension strength ratio at inflection point ($f\_XT = 0.1$) and proportion of G_XT dissipated before the inflection point ($f\_GT = 0.4$) can be assumed.
- Fracture toughness in longitudinal compression (G_XC):
  - Reverse engineer the value of the fracture toughness in axial compression (G_XC) by matching the open hole strength in compression of a quasi-isotropic layup. This could be set up as a project in Digimat VA by varying the G_XC value.
  - As a starting value, G_XC value for AS4/8552 in the Digimat VA database (106.3 mJ/mm^2) could be used.

- The Digimat VA default values for the longitudinal compression strength ratio at inflection point ($f\_XC = 0.1$) and proportion of G_XC dissipated before the inflection point ($f\_GC = 0.4$) can be assumed.

- Fracture toughness in transverse tension (G_YT):
  - The fracture toughness in transverse tension may be assumed to be the same as the interlaminar fracture toughness in mode I (as defined in the delamination model definition)
- Fracture toughness in in-plane shear (G_SL):
  - The fracture toughness in in-plane shear may be assumed to be the same as the interlaminar fracture toughness in mode II (as defined in the delamination model definition)
- Fracture angle (α_0):
  - The fracture angle has a default value in Digimat VA of 0.925 radians (53 degrees). This recommendation comes from the paper by Furtado et al. (2019).
- Fracture toughness in transverse compression (G_YC):
  - The fracture toughness in transverse compression may be estimated using the formula $G\_YC = G\_SL/\cos(\alpha_0)$, where G_SL is the Fracture toughness in in-plane shear and α_0 is the fracture angle.

Custom Material Model

Custom material models can be used instead of Digimat material models. The custom material model has to be implemented in a shared library.

In Digimat-VA, custom material models first have to be defined in the Customization window (which can be accessed from the Digimat-VA main menu), see Figure 3-11. A list of available custom material models is displayed in the left part of Figure 3-11. Using the buttons below the list of available models, it is possible to

- Create a new custom material model
- Delete an existing custom material model
- Import/export custom material models. Custom material models are exported to self-contained `.xml` files. Those `.xml` files also contains the shared library files (both Windows DLL and Linux versions), which makes possible exchanging Digimat-VA custom material models between different machines as easy as the transfer of a single file.

The right part of the window in Figure 3-11 allows to fully define the selected custom material model. The following parameters have to be defined

- Name: a unique name identifying the custom material model
- Dynamic libraries: full path to the Windows DLL and (optionally) to the Linux .so file containing the custom material model implementation

- Input parameters: specify the complete list of input parameters requested by the custom material model implemented in the shared library. Note that the order of definition is important: it must be the same as the order in which the custom material model implementation expects to receive its input parameters. For each parameter, it is possible to define a name, the dimension (necessary for unit conversions), and the variability level (if any).
- Tension-compression differentiation

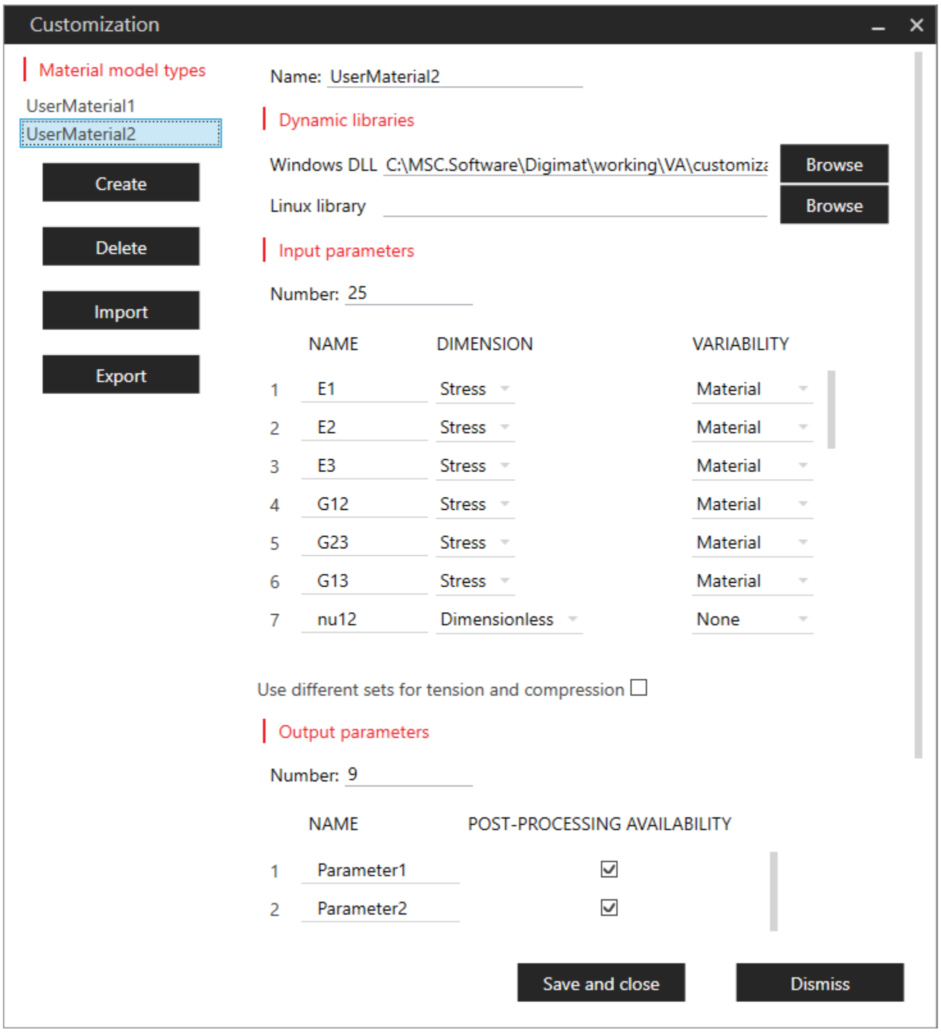


Figure 3-11 Material model customization window.

- Output parameters: the list of output parameters of the custom material model. For each output parameter, it is possible to specify whether or not it should be included in the FEA output files (and therefore, whether or not it will be available for post-processing inside Digimat-VA).

The step of defining the custom material model only has to be performed once. All custom material models are saved in a permanent way, making it possible to reuse them in different sessions of Digimat-VA.

The usage of a custom material model is very similar to a classical Digimat material model. The main differences are

- No calibration is available for custom material models. The user has to provide the required input parameters at the right volume fraction and for the right environmental conditions. See Figure 3-12.
- The material behavior plot in Figure 3-12 internally uses a mono-element (instead of a simple Digimat run with Digimat material models). Therefore, the refresh speed of the plot is much slower and plot refresh is not performed automatically (only through manual click on the **Refresh** button).

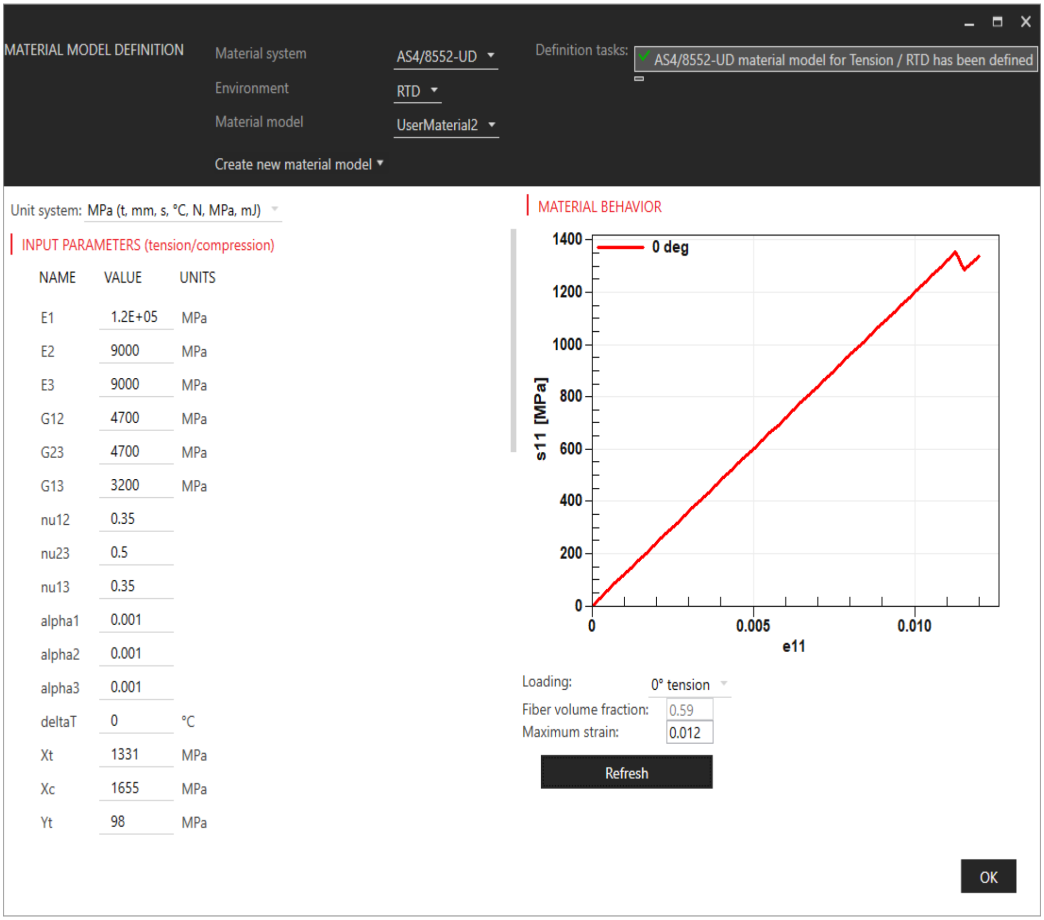


Figure 3-12 Custom material definition window.

Custom material in Digimat-VA must be written in Fortran language and linked as a dynamic library (`.dll` file under Windows platform and `.so` file under Linux platform).

The function that will be called by Digimat-VA to describe user material behavior must be called `usermaterial`. Declaration of `usermaterial` function must be defined as:

```
usermaterial(stiff, stifft, strain, dstrain, stress,
  & statev, dstatv, ngens, elemid, ipid, layerid,
  & matus, ndi, nshr, disp, dispt,
  & coord, ff0, frot0, stretch0,
  & eigv0, ff1, frot1, stretch1,
  & eigv1, ncrd, itel, ndeg,
  & nnode, jtype, lclass, ifr, ifu,
  & nstats, matname, isunit, nid, nrd, ncd,
  & idata, rdata, cdata,
  & lovl, returncode,
```

```
& time, dtime,
& celent, newdt, kspt, temp, dtemp, kstep, kinc)
!DEC$ ATTRIBUTES DLLEXPORT :: usermaterial
```

Command `!DEC$ ATTRIBUTES DLLEXPORT::usermaterial` is mandatory to ensure that `usermaterial` symbol will be exported and can be found by Digimat-VA.

Input and output variables of user material model are described in Table 3-1, 3-2 and 3-3.

Table 3-1 Input variables for Digimat-VA user defined material model (1).

`strain`	Mechanical strain tensor at increment n, ndi+nshear size.
`dstrain`	Mechanical strain tensor increment at increment n+1, ndi+nshear size.
`statev`	State variables at increment n, nstats size.
`ngens`	Size of the stress-strain law.
`elemid`	Index of current element.
`ipid`	Index of current integration point
`layerid`	Index of current layer. Not used in Digimat-VA 2023.3.
`matus`	User material identifier.
`ndi`	Number of direct components (3 for solid elements).
`nshear`	Number of shear components (3 for solid elements)
`disp`	Incremental displacements.
`dispt`	Displacements at increment n (at assembly lovl = 4) and displacements at increment n+1 (at stress recovery lovl = 6).
`coord`	Coordinates of the nodes of current element.
`ff0`	Deformation gradient at the beginning of the increment.
`frot0`	Rotation tensor at the beginning of the increment
`strech0`	Square of principal stretch ratios at the beginning of the increment.
`eigv0(i,j)`	I principal direction components for J eigenvalues at the beginning of the increment.
`ff1`	Deformation gradient at the current increment.
`frot1`	Rotation tensor at the current increment
`strech1`	Square of principal stretch ratios at the current increment.
`eigv1(i,j)`	I principal direction components for j eigenvalues at the current increment
`ncrd`	Number of coordinates.
`itel`	Dimension of ff and frot; 2 for plane-stress and 3 for the rest of the cases.
`ndeg`	Number of degrees of freedom.
`nnode`	Number of nodes per element.

Table 3-1 Input variables for Digimat-VA user defined material model (1).

`t`	Time at increment n.
`dt`	Time increment of increment n+1.
`nnodes`	Number of nodes of current element
`jtype`	Element type.
`lclass(1)`	Element class. Not used in Digimat-VA 2023.3
`ifr`	Set to 1 if R has been calculated.
`ifu`	Set to 1 if strech has been calculated.
`nstats`	Number of state variables.
`matnamec`	Material name.
`isunit`	Parameter indicating the unit system. Not used in Digimat-VA 2023.3

The order of definition of strain and stress tensor is defined as 11, 22, 33, 12, 23, 13.

Return code must be defined as:

Table 3-2 Input variables for Digimat-VA user defined material model (2).

`nid`	Number of auxiliary integer numbers. Not used in Digimat-VA 2023.3.
`nrd`	Number of auxiliary real numbers.
`ncd`	Number of auxiliary character strings. Not used in Digimat-VA 2023.3.
`idata`	Array with auxiliary integer numbers. Not used in Digimat-VA 2023.3.
`rdata`	Array with auxiliary real numbers. It contains complete list of input parameters given by user in Digimat-VA GUI.
`cdata`	Array with auxiliary character strings. Not used in Digimat-VA 2023.3.
`lovl`	Set to 4 for assembly phase and to 6 for stress recovery phase.
`time`	Current time.
`dtime`	Time increment.
`celent`	Characteristic length of element.
`kspt`	Loadcase number.
`temp`	Temperature at start of increment. Not used in Digimat-VA 2023.3.
`dtemp`	Temperature increment. Not used in Digimat-VA 2023.3.
`kstep`	Time step number.
`kinc`	Increment number.

Table 3-3 Output variables for Digimat-VA user defined material model

`stiff`	Stress strain law to be formed, ngens*ngens size.
`stifft`	Change in stress due to temperature effects, ngens size. Not used in Digimat-VA 2023.3.
`stress`	Stress to be updated by user, ndi+nshear size.
`dstatev`	Increment of state variables.
`returncode`	Return code of user subroutine (see below for allowed values).
`newdt`	New time increment. Not used in Digimat-VA 2023.3.

- 0 if no error and no action required.
- 1 if error at input level (e.g., bad value of parameter).
- 2 if computation must stop immediately after increment completion.
- 3 if time step must be divided by a factor 2.0.

In order to compile Fortran user subroutine, export usermaterial symbol and build the user material dynamic library, a Fortran compiler must be installed. Digimat-VA 2023.3 supports Intel Fortran 19.0. Compile command is given by:

```
ifort.exe /dll /def:usermaterial -o usermaterial.dll
usermaterial.for
```

Material Variability

Upon successful calibration of the material model, the material variability can be defined in the **Digimat material model** tab (see Figure 3-4). The list of material parameters supporting variability is shown in Figure 3-13 for a UD material and in Figure 3-14 for a woven material. In that list, it is possible to select which material parameters exhibits variability and which don't.

The modeling assumption for material variability in Digimat-VA is that each varying material parameter follows a normal law. The variation of a material parameter can therefore be defined by a single dimensionless parameter, the coefficient of variation, which is the ratio between the standard deviation and the mean value of the material parameter. The only material parameter that doesn't follow this rule is the Fiber alignment range. Since its mean value is obviously 0, we directly specify the standard deviation (in degrees) for that material parameter.

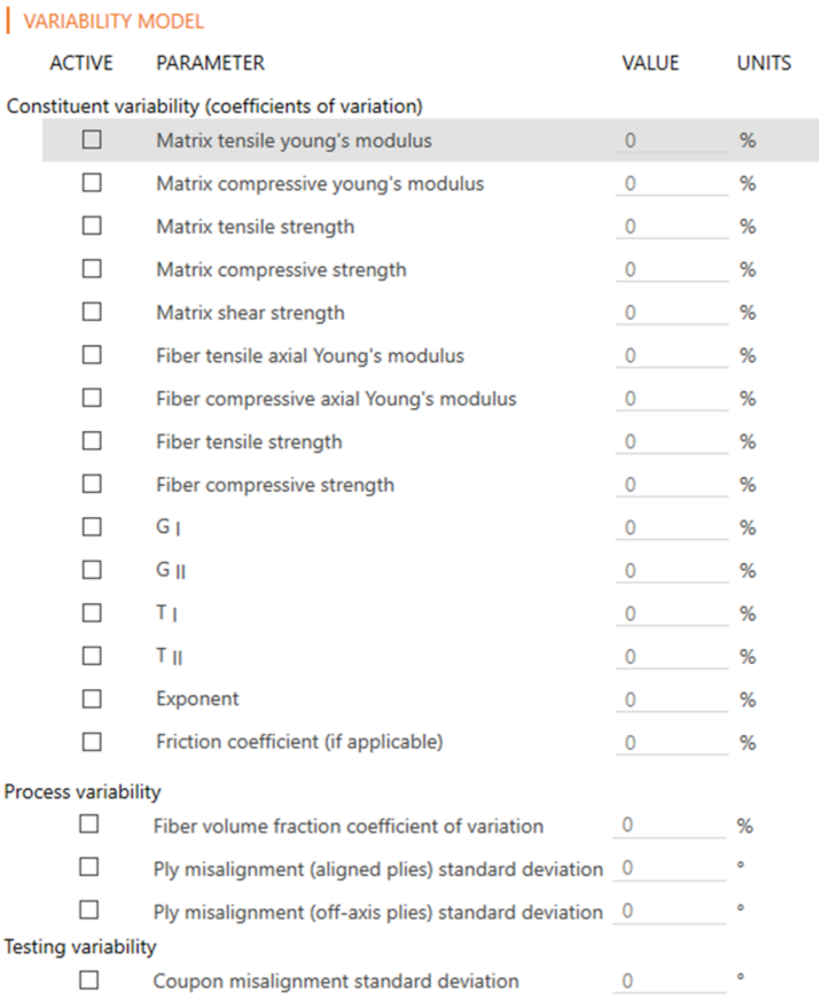


Figure 3-13 Definition of material variability in a Digimat UD material model.

VARIABILITY MODEL

ACTIVE	PARAMETER	VALUE	UNITS
Constituent variability (coefficients of variation)			
☐	Matrix tensile young's modulus	0	%
☐	Matrix compressive young's modulus	0	%
☐	Matrix shear strength	0	%
☐	Warp tensile axial Young's modulus	0	%
☐	Warp compressive axial Young's modulus	0	%
☐	Warp tensile strength	0	%
☐	Warp compressive strength	0	%
☐	Weft tensile axial Young's modulus	0	%
☐	Weft compressive axial Young's modulus	0	%
☐	Weft tensile strength	0	%
☐	Weft compressive strength	0	%
☐	G_I	0	%
☐	G_{II}	0	%
☐	T_I	0	%
☐	T_{II}	0	%
☐	Exponent	0	%
☐	Friction coefficient (if applicable)	0	%
Process variability			
☐	Fiber volume fraction coefficient of variation	0	%
Testing variability			
☐	Coupon misalignment standard deviation	0	°

Figure 3-14 Definition of material variability in a Digimat woven material model.

FE Analysis

The FE analysis screen (see Figure 3-15) allows to control the creation of FE analyses for all virtual tests. More specifically, this step provides control over three different aspects:

- the definition of FEA settings such as applied strain, mesh size, number of time steps
- the choice of output for the FEA results such as stress, strain or damage fields
- for parametric studies, the definition of parameter variations
- the choice of solver to be used

The description of these aspects is detailed in the next sections.

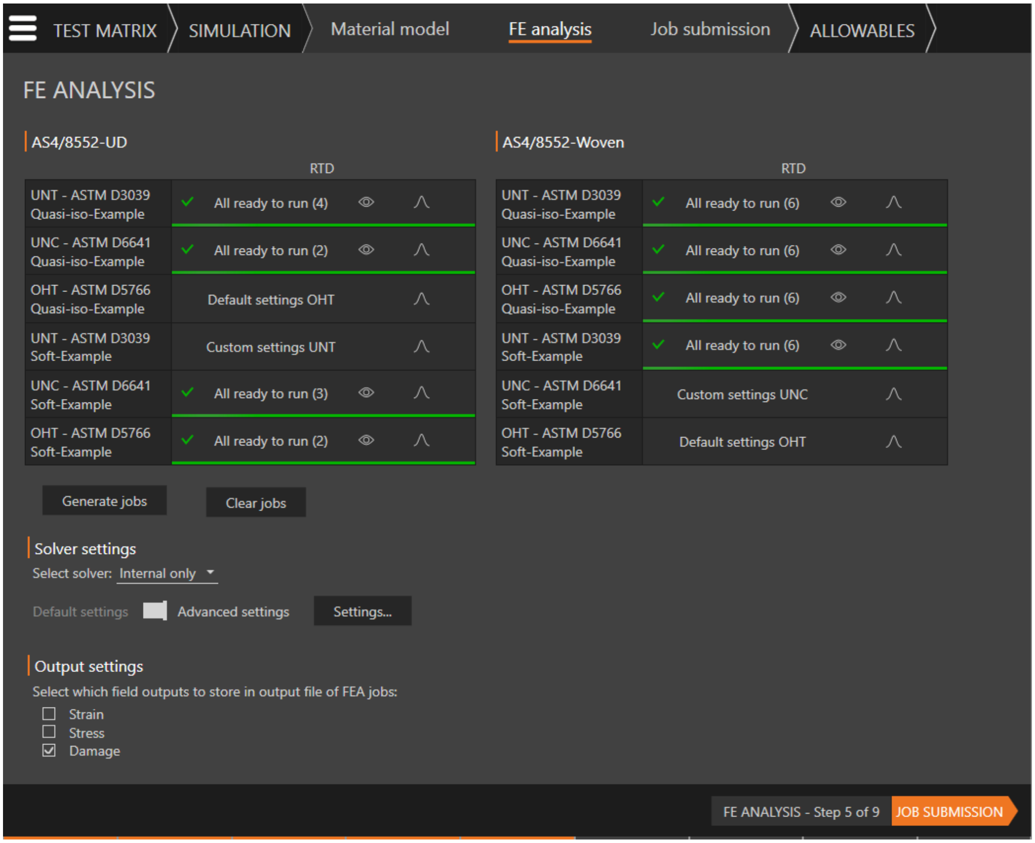


Figure 3-15 FE analysis screen.

Solver Selection

Four solver options are available (see Figure 3-16):

- Internal only: will only use of Digimat-VA's internal solver
- Favor internal: will use Digimat-VA's internal solver where it is supported and the external solver in the other cases
- External only: will only use the external solver
- Custom: will use the solver selected in the advanced mesh settings

Currently Digimat-VA only supports Abaqus Explicit as an external solver.

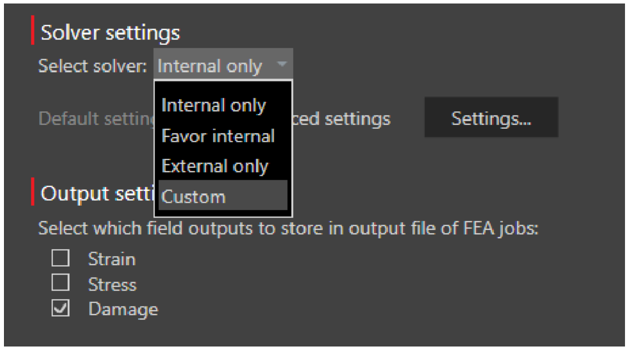


Figure 3-16 Solver selection options.

FEA Settings

The FEA settings are specific to the type of test. Besides, the FEA settings for Advanced PFA models, clearly identified by the (**Advanced PFA)** tag in their name, contain a few more items than the FEA settings which can be used with the FPF and the Standard PFA models.

Initially, default settings are used for all virtual tests. The settings can be changed by switching the toggle button at the bottom of the screen to **Advanced settings**, which allows defining new settings and assigning specific settings to individual virtual tests. Custom settings can be assigned to virtual tests using the right click context menu of the test matrices items.

Most of the first settings are available for all tests (see Figure 3-17).

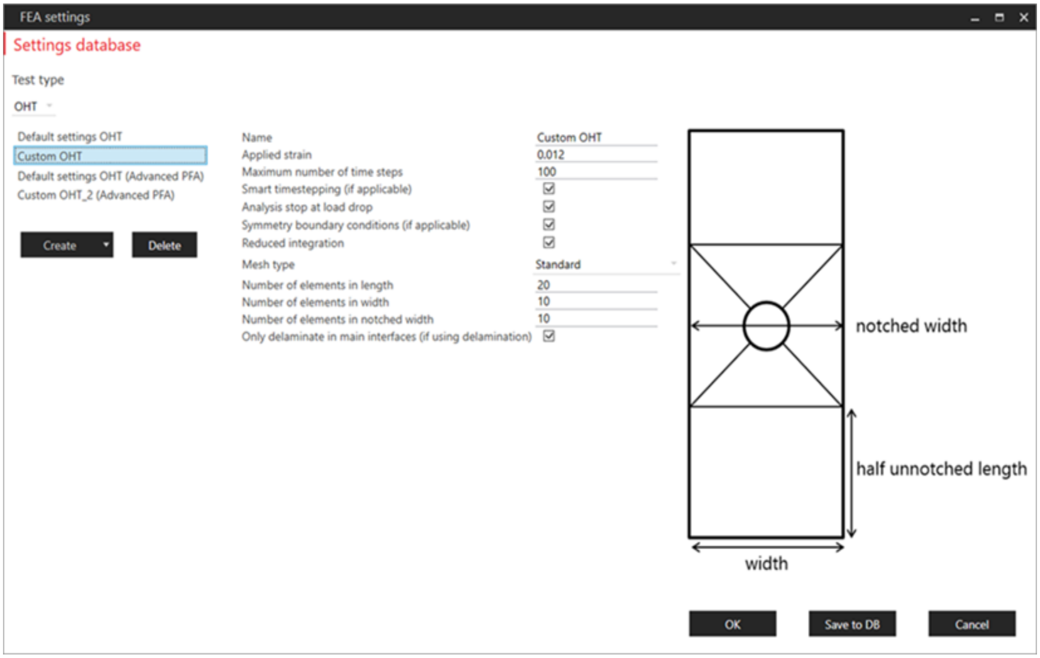


Figure 3-17 FEA settings window.

- Name
- Applied strain: This setting defines the applied displacement according to the coupon length. It is important to specify a value of strain high enough to ensure that failure is reached, but not too high to avoid unnecessary computational cost. For bearing tests, it does not strictly correspond to the bearing strain, defined after a relative displacement (refer Allowables and Composite Characterization). To ensure a consistent order of magnitude, it consists in the ratio between the applied displacement and the hole diameter (twice the hole diameter for 2-piece tests).
- Maximum number of time steps: This setting defines the strain increment from the applied strain. The larger that value, the larger the number of increments in the simulation. When using the FPF or the Standard PFA models, the results will be saved at all increments. When using the Advanced PFA model, results will only be saved at maximum 200 increments uniformly spread over the entire simulation to avoid creating prohibitively large result files.
- Smart time stepping: If checked, the strain corresponding to damage initiation is estimated after the first time step; the second time step is adapted to reach this strain in one step. Then, the strain increment is equal to the applied strain divided by the number of time steps. This setting is not applicable to filled hole and bearing tests involving contact nonlinearities.

- Analysis stop at load drop: This setting lets Digimat-VA stop the analysis when the global stress decreases significantly after a maximum.
- Symmetry boundary condition: When symmetric layups are used, only one half of the plies will be modelled (the upper half). A symmetry boundary condition will be applied. This setting has no effect with non-symmetric layups, filled hole, bearing and bending tests.
- Reduced integration: If checked, linear hexahedral elements with reduced integration (i.e. one single integration point) will be used. Otherwise, fully integrated hexahedral elements are used.
- Number of elements in coupon length
- Number of elements in coupon width
- Only delaminate in main interfaces (if using delamination): If unchecked, cohesive elements are added at the interface between all plies making it possible for all interfaces to delaminate. If checked, cohesive elements are only added at the interface between the plies with dissimilar orientations.

For unnotched tests, the following extra setting is available regardless of the type of material modeling strategy. Unnotched coupon modeling approach. It allows to choose between two options

- Full coupon mesh: the default approach, especially recommended if progressive failure is being used
- Mono-element: instead of modeling a full coupon, only one element per ply is used, coupled with periodic boundary conditions. This option is mainly interesting with the First Ply failure modeling strategy, as it allows to retrieve the same results as the CLT (Classical Laminate Theory). It is not recommended to use it with progressive failure.

Still for unnotched tests but only when using the Advanced PFA models, the following extra settings are available:

- Damage zone ratio, which is the ratio of the length where damage is allowed to occur to the free length. This gives the possibility to reduce even further the zone over which the material is allowed to damage and, together with the other parameter below, to have a better control over the element size in that zone.
- Number of elements in the damage zone, which specifies how many elements out of the total number of elements in length will span the length of the damage zone.

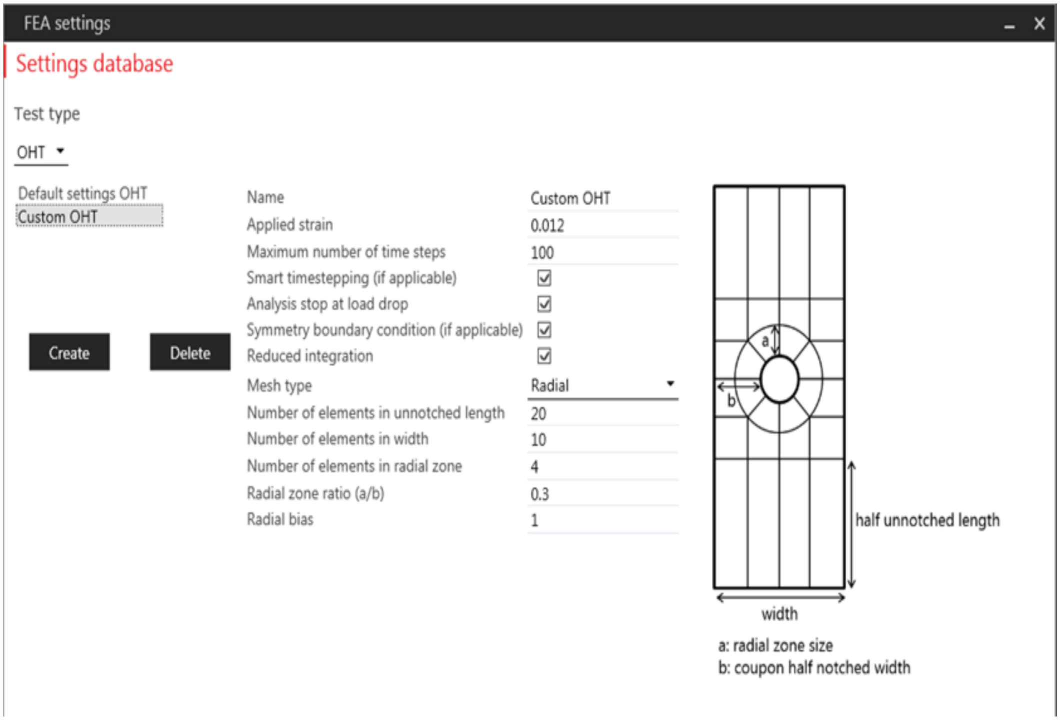


Figure 3-18 FEA settings window for an open hole test with radial mesh.

For open hole tests, when using the FPF or the Standard PFA models, the user can choose between two types of meshes: standard (see Figure 3-19) or radial (see Figure 3-20).

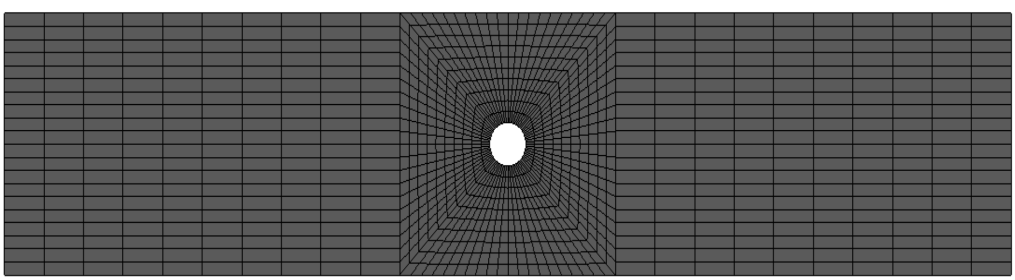


Figure 3-19 Standard mesh for open-hole coupon.

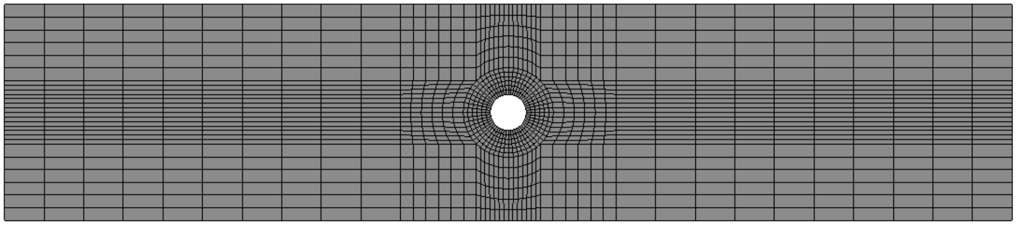


Figure 3-20 Radial mesh for open-hole coupon.

Both types come with their own extra FEA settings:

- Number of elements in coupon notched width for the standard mesh type
- For the radial mesh type (see Figure 3-18):
  - Number of elements in radial zone: number of elements along a radius of the radial zone. The circumferential size is also deduced from this parameter, in order to have elements as square as possible.
  - Radial zone ratio: ratio of the distance between coupon edge and hole edge to the distance between hole edge and boundary of the radial zone.
  - Radial bias: This setting allows to have non uniform element size in the radial zone. It is defined as the ratio between the size of the largest element (last element of the radial zone, furthest away from the hole) to the size of the smallest element (close to the edge of the hole). Therefore a value of 1 leads to a uniform mesh in the radial zone.

For open hole tests, when using the Advanced PFA models, only the radial mesh type is available but it offers two extra settings compared to the radial mesh for FPF and Standard PFA models:

- Bias type: arithmetic or geometric, which controls how the element sizes evolve in the biased region. With an arithmetic progression, a fixed size increment is added from one element to the other: l, $l+\Delta l$, $l + 2\Delta l$, etc. With a geometric progression, a fixed scaling factor s is applied from one element to the other: l, sl, s^2l, etc.
- Aspect ratio of the elements around the hole, which defines the ratio of the element size in the radial direction to the element size in the circumferential direction for the elements around the hole

Note:	The values provided for the number of elements in length, width and notched width are only considered as indicative: they might be slightly adjusted to avoid creating distorted elements.

Besides their exclusive radial mesh definition, filled hole and bearing tests require additional settings related to the fastener preload and its interaction with the coupon (see Figure 3-21).

- Friction coefficient
- Number of time steps for torquing
- Safety factor for torquing temperature map. This is an advanced user parameter which can be increased if the full fastener torque is not achieved at the end of the torquing step.
- Contact discretization: According to this setting, the FE solver enforces contact conditions on individual nodes (node-to-surface) or in an average sense (surface-to-surface).
- Ratio between contact tolerance and ply thickness: This setting influences the distance until which nodes are considered touching a surface and not yet penetrating.

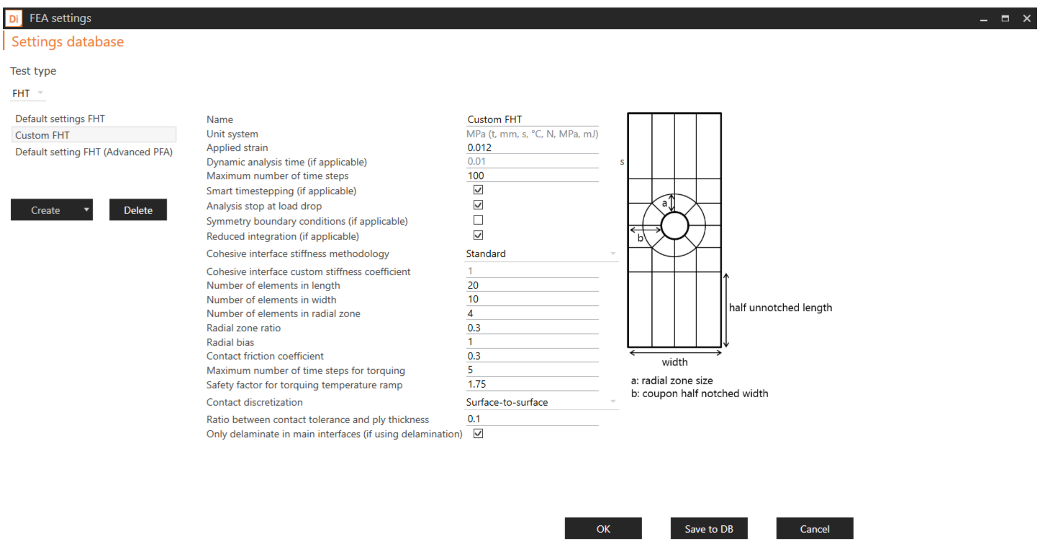


Figure 3-21 FEA settings window for a filled hole test.

The friction coefficient also needs to be provided for bending, short-beam strength, end notched flexure, drop weight impact and tension/compression after impact tests.

For drop weight impact and tension/compression after impact tests, there are the following additional settings (see Figure 3-22 and Figure 3-23):

- Impact step duration (defines the total time of the impact event).
- Clamping step duration (defines the analysis time for applying the clamping load).
- Use thick shell elements (allows for the selection between continuum elements and continuum shell elements).
- Use element deletion (activation of element deletion upon reaching full damage in the plies).

- Use cohesive element deletion (activation of element deletion upon reaching full damage in the cohesive elements).
- Element size (in non-damage area): Specified in distance units.
- Element size (in damage area): Specified in distance units.
- Impactor additional distance (to coupon top surface): Only available for Advanced PFA models to account for high levels of warpage during pre-impact cooldown simulations.
- Clamps additional distance (to coupon top surface): Only available for Advanced PFA models to account for high levels of warpage during pre-impact cooldown simulations.
- Pre-clamping force factor: Only available for Advanced PFA models to ensure that the clamps remain in contact with the laminate during pre-impact cooldown simulations.

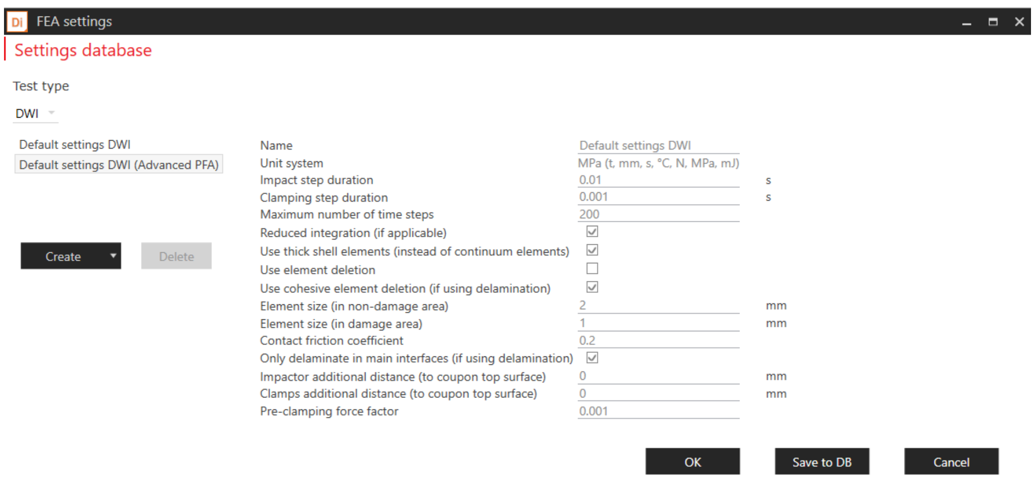


Figure 3-22 FEA settings for Drop Weight Impact tests.

Additionally, for Tension/Compression After Impact tests, the following additional settings are available (with respect to the tension/compression after impact settings), see Figure 3-23.

- Damping step duration: after the impact step and before the tension/compression step, a damping step is inserted so that the vibrations in the coupon generated by the impact can smooth out. This parameter controls the duration of that damping step.
- Loading step duration: the duration of the tension/compression step, performed after the damping step.
- Analysis stop at rebound: if checked, the impact step is stopped as soon as the impactor has lost contact with the plate after the impact.

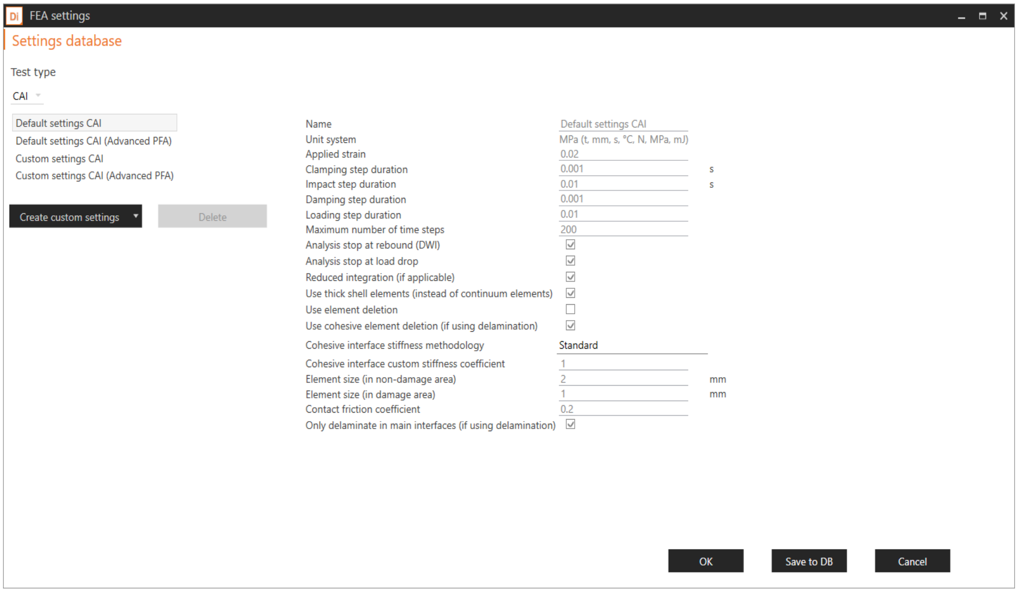


Figure 3-23 FEA settings for Compression After Impact test.

If the **custom** option was selected in the Solver selection, an additional option is available in the FE settings to choose which solver should be used (see Figure 3-24).

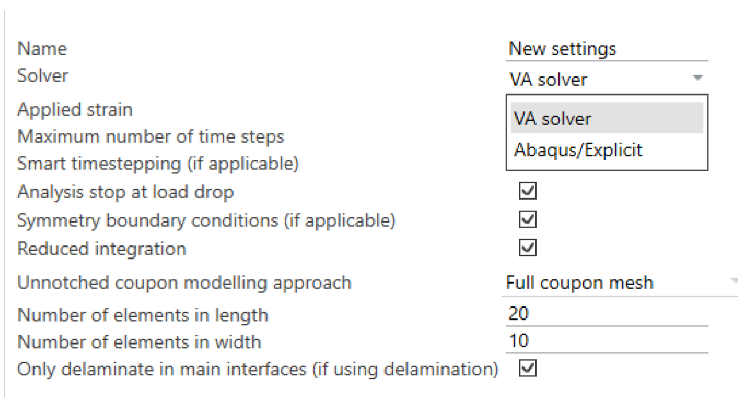


Figure 3-24 FEA settings for a Filled Hole test.

For the curved beam strength test, there are the following additional settings (see Figure 3-25):

- Applied displacement (vertical displacement of indentors in model unit system)
- Number of elements in curvature radius
- Number of elements in width

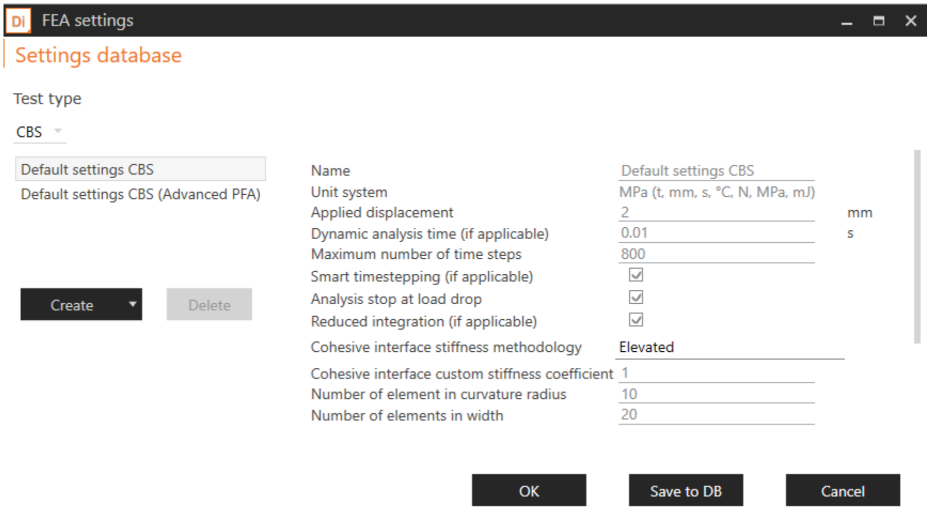


Figure 3-25 FEA settings for Curved-beam Strength Model.

For the DCB and C-ELS test, there are the following additional settings (see Figure 3-26):

- Applied displacement: Vertical displacement of the loading block (half the opening displacement)
- Number of elements in length: Number of elements along the specimen length excluding the damage zone
- Number of elements in the damage zone: Number of elements along the specimen damage zone where cohesive damage is activated
- Only delaminate in main interfaces:
    - Activated: Only delamination at the mid thickness where the crack will propagate
    - De-activated: All interfaces have delamination activated

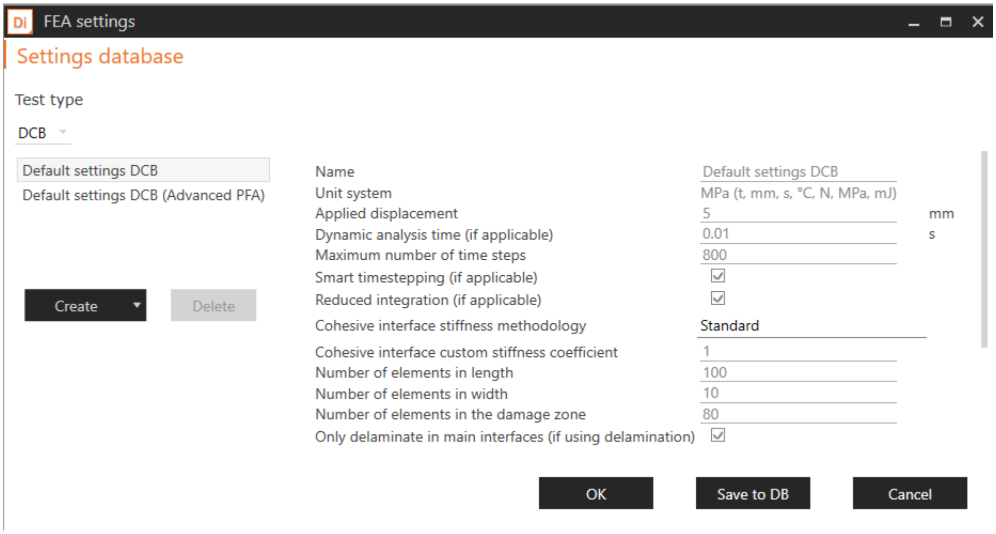


Figure 3-26 FEA settings for DCB and C-ELS test.

FEA Outputs

The choice of field outputs that will be stored in the FEA output files affects the type of post-processing that will be allowed as well as the size of the output file generated. De-selecting all outputs allows a minimal output file size, but no local post-processing will be possible, only the macroscopic stress-strain curve will be accessible (and therefore also the stiffness and strength).

The default is to request output of the 3 main damage variables (or failure indicator variable if the First Ply Failure modeling strategy is being used) related to the longitudinal (or warp) and transverse (or weft) directions as well as shear (see section Damage Evolution Laws for details about the meaning of these outputs).

Parametric Study Definition

When running a parametric study in Digimat-VA, the exact definition of the parameters variation happens at the FE analysis substep. Each test configuration is treated separately and can be managed by clicking on the dedicated icon in the test box. This opens the parametric study definition window which allows to control:

- Strategy for parametric study: user can choose between single parameter matrix or full cross matrix strategy. The first strategy explores parameter variations independently of the other active parameters, while the second strategy combines all parameter variations altogether. With the full cross matrix strategy, the number of tests to be run can become significant.

- Material parameters: user can vary the desired constituent stiffness and strengths as well as the fiber volume fraction
- Layups parameters: user can study the effect of layup misalignment, whether applied for the whole coupon, only for 0 degree plies, or only for 90 degree plies.
- Test parameters: if relevant for the selected test, user can study the influence of coupon length, width as well as hole diameter.

Parametric study parameters

ACTIVE	PARAMETER	NOMINAL VALUE	UNITS	VARIATIONS
Material				
☐	Matrix Tension Young's modulus	4667.7	MPa	
☐	Matrix Tensile strength	56.118	MPa	
☐	Matrix Compressive strength	231.67	MPa	
☐	Matrix Shear strength	62.074	MPa	
☐	Fiber Tension Axial Young's modulus	2.1769E+05	MPa	
☐	Fiber Tension In-plane Young's modulus	15236	MPa	
☐	Fiber Volume fraction	0.59		
☐	Fiber Tensile strength	3413.1	MPa	
☐	Fiber Compressive strength	2366.2	MPa	
☐	G_{I} (fracture toughness in mode I)	0.28	mJ/mm^2	
☐	G_{II} (fracture toughness in mode II)	0.79	mJ/mm^2	
☐	T_{I} (normal strength in mode I)	26	MPa	
☐	T_{II} (normal strength in mode II)	78.1	MPa	
☐	Exponent	1.45		
☐	Friction coefficient (if applicable)	0.2		
Layup				
☐	Ply misalignment (whole coupon)	0	°	
☐	Ply misalignment (aligned plies)	0	°	
☐	Ply misalignment (off-axis plies)	0	°	
Test				
☐	Length	254	mm	
☐	Width	25.4	mm	

Figure 3-27 Definition of a parametric study for a given test.

To activate a parameter in the parametric study, the parameters needs to be toggled on. The parameter variation must then be defined. Two definitions of parameter variations are possible:

- A semicolon separated list of discrete values (e.g. 40;50;60;70)
- A range with the notation min:max:increment (e.g. 40:70:10)

In addition to the parameter set defined in the parametric study definition window, a reference case corresponding to the nominal values of the parameters is added. Note that the parametric study definition is specific to each test configuration and must therefore be defined on a case-by-case basis.

Defect Study Definition

When variability type is set to Defect study in the Variability tab of Digimat-VA (see Variability definition), the desired set of defects can be defined at this stage, before generating the FEA jobs. Each virtual test can be assigned one or several defects, independently from the other virtual tests. For each virtual test with defect, two FEA jobs will be generated, one with the selected defect(s) and another one without any defect (pristine).

The KDF (knock-down factor) will then be computed as the ratio of the result on the coupon with defect divided by the same result on the pristine coupon. KDF are computed for strength and stiffness (only for unnotched coupon). The following type of defects are supported:

- waviness
- intraply porosity
- interply porosity
- initial delamination
- AFP gap

Waviness is always defined on the whole width of the coupon and is affecting a portion of the length, determined by the waviness wavelength. Other types of defect can be assigned either to a specific and limited area of the coupon (determined by a length and a width and the x and y coordinates of the center of the defect area) or globally to the whole coupon.

It is possible to have multiple defects of the same type on a single coupon, provided that they do not overlap (this is automatically checked when validating the input in the defect definition window). It is not possible to combine different type of defects in the same coupon.

Waviness

The location of a waviness defect is defined by the position of the center of the waviness (along the coupon length) and by its wavelength and orientation. When applied on an unnotched coupon, it is possible to define a mesh refinement factor to use a more refined mesh inside the wavy area. The elements inside the wavy area are then subdivided by this factor in the coupon length direction. The amplitude of the waviness can be defined in 5 different ways

- Uniform (same amplitude at all interfaces)
- Hump (amplitude gradually increasing from 0 to max from bottom to top interface)
- Indentation (amplitude gradually increasing from 0 to max from top to bottom interface)

- Embedded (amplitude evolves bilinearly from 0 at the bottom interface to max at the middle interface to 0 at the top interface)
- Custom

Whatever the type of waviness, for elements in the wavy area, material orientation is always adjusted in order to follow the ply undulation.

When the waviness leads to a varying thickness in a given ply, the local fiber volume fraction is adjusted in that area to account for the fact the local extra volume is filled by resin while the local amount of fiber remains constant. Therefore, depending on the waviness type, not all values of maximum amplitude are acceptable, as some values may lead to negative ply thickness and / or fiber volume fraction larger than 1. Those two conditions are automatically checked by Digimat-VA before the defect can be assigned.

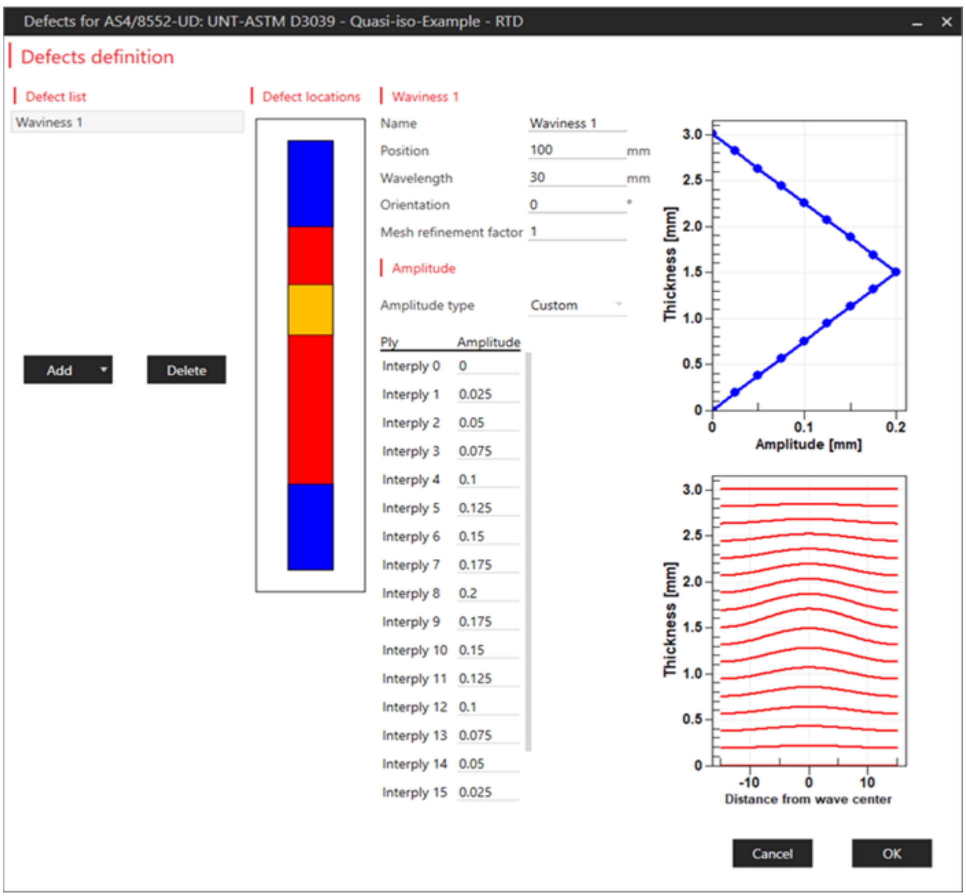


Figure 3-28 Definition of a waviness defect (custom amplitude type).

Intraply Porosity

To model the effect of intraply porosity on the material properties, 5 different RVE models will be generated and solved using Digimat-FE (for longitudinal tension and compression, transverse tension and compression and in-plane shear). Those RVE models will allow to compute the porous mechanical ply properties. Based on those porous mechanical properties, a new Digimat material model will be calibrated and assigned to the porous area of the coupon. The parameter Porosity fraction is the volume fraction of porosity present in the composite.

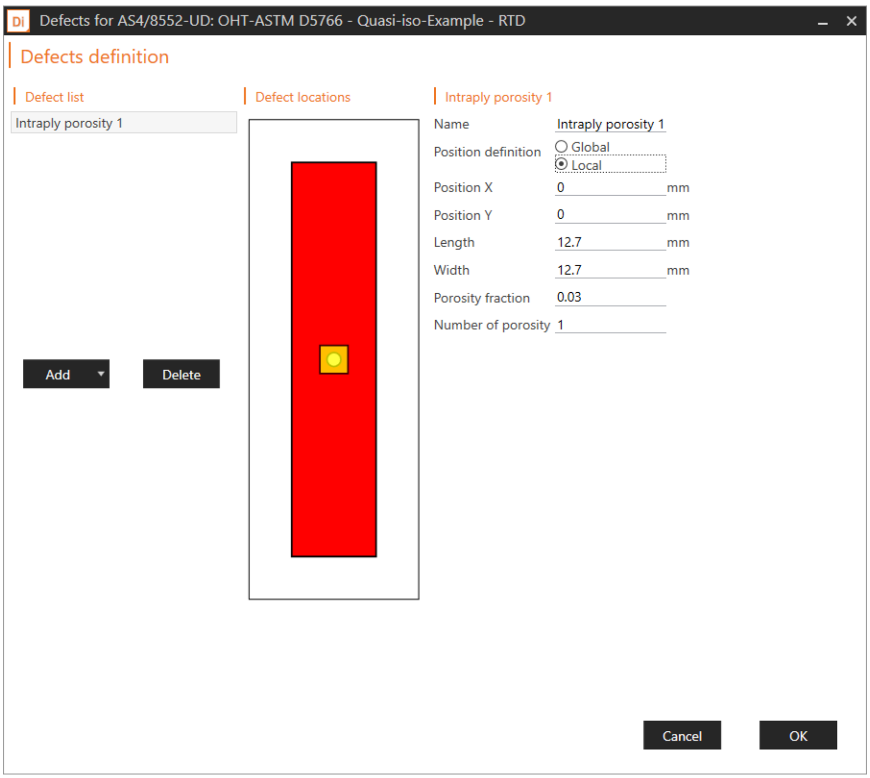


Figure 3-29 Definition of an intraply porosity defect.

This calibration process starts at the beginning of the FEA job generation and can be monitored from the porosity calibration window (see Figure 3-30). Once all RVE model simulations are completed, it is possible to compare the ply properties and the material models for the porous and pristine material in this window.

It is also possible to view the stress-strain curves of each individual rve analysis by clicking on the curve icon in the table of the porosity calibration window (see Figure 3-30). When generating the FEA models, the calibrated porous material model will be assigned to all ply elements belonging to the porous area, while the pristine material model will be assigned to all other elements.

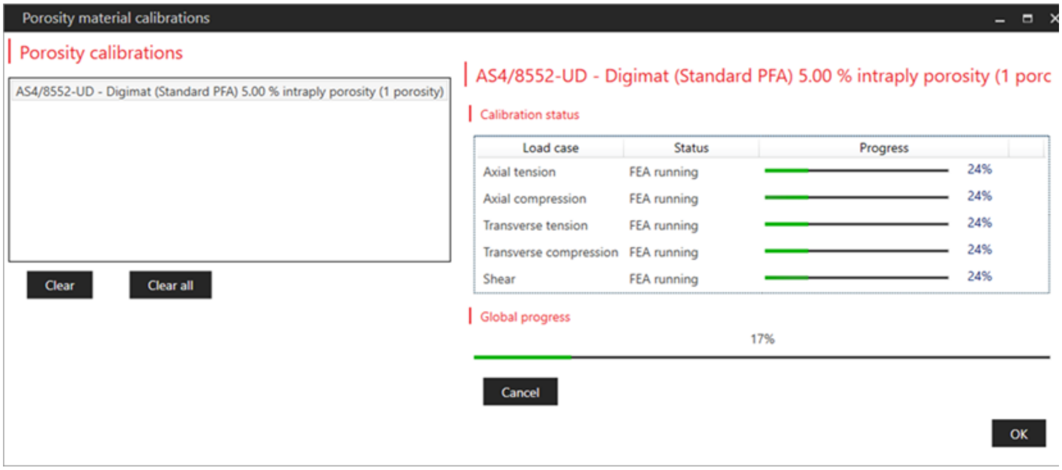
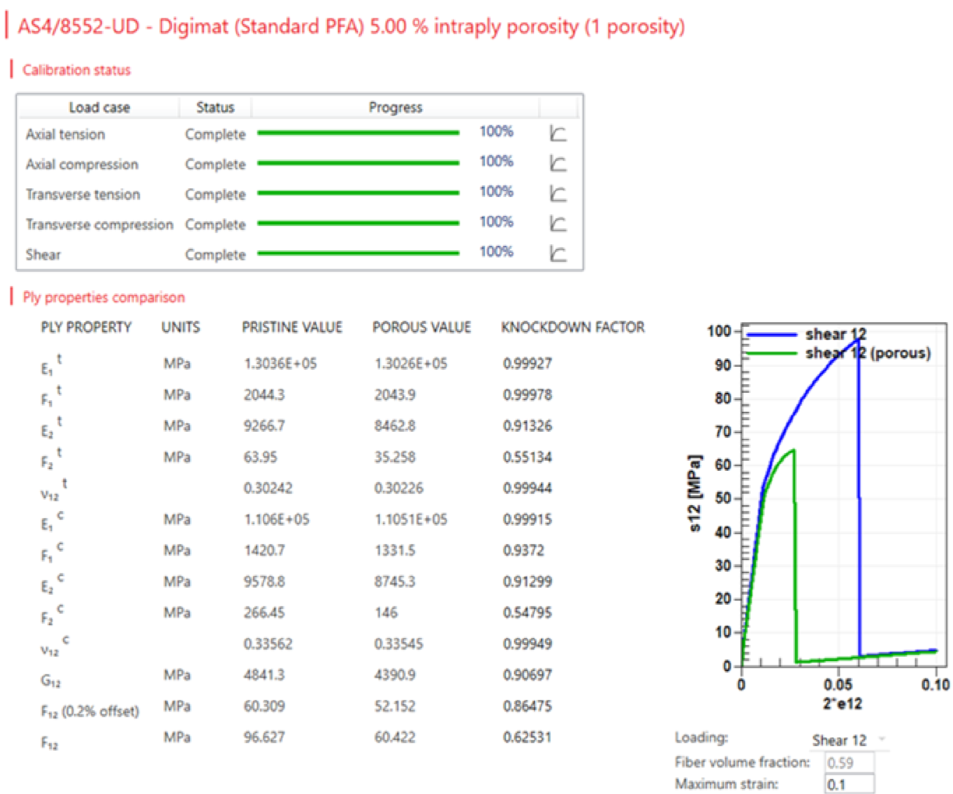


Figure 3-30 Porous material calibration running.



PLY PROPERTY	UNITS	PRISTINE VALUE	POROUS VALUE	KNOCKDOWN FACTOR
E_1^t	MPa	1.3036E+05	1.3026E+05	0.99927
F_1^t	MPa	2044.3	2043.9	0.99978
E_2^t	MPa	9266.7	8462.8	0.91326
F_2^t	MPa	63.95	35.258	0.55134
ν_{12}^t		0.30242	0.30226	0.99944
E_1^c	MPa	1.106E+05	1.1051E+05	0.99915
F_1^c	MPa	1420.7	1331.5	0.9372
E_2^c	MPa	9578.8	8745.3	0.91299
F_2^c	MPa	266.45	146	0.54795
ν_{12}^c		0.33562	0.33545	0.99949
G_{12}	MPa	4841.3	4390.9	0.90697
F_{12} (0.2% offset)	MPa	60.309	52.152	0.86475
F_{12}	MPa	96.627	60.422	0.62531

Figure 3-31 Porous material calibration completed.

Interply Porosity

Interply porosity is modelled in a way very similar to intraply porosity. The main difference with intraply porosity is that it affects both the ply and the interface properties. It can therefore only be applied to coupons with interface delamination is allowed. Another difference with interaply porosity is that two different options are available to estimate the knock-down factors on the ply and interface properties: user input and calibration based on porosity.

The user input allows to directly specify the knock-down factors for the different ply and interfaces properties. The calibration based on porosity will create different RVE models (7 in this case, 5 for the ply properties, as for the intraply porosity, and 2 for the interface properties, in normal and shear mode) using Digimat-FE. Similarly to the intraply porosity, those RVE models will be solved and will allow to compute the porous mechanical properties for the ply and the interface.

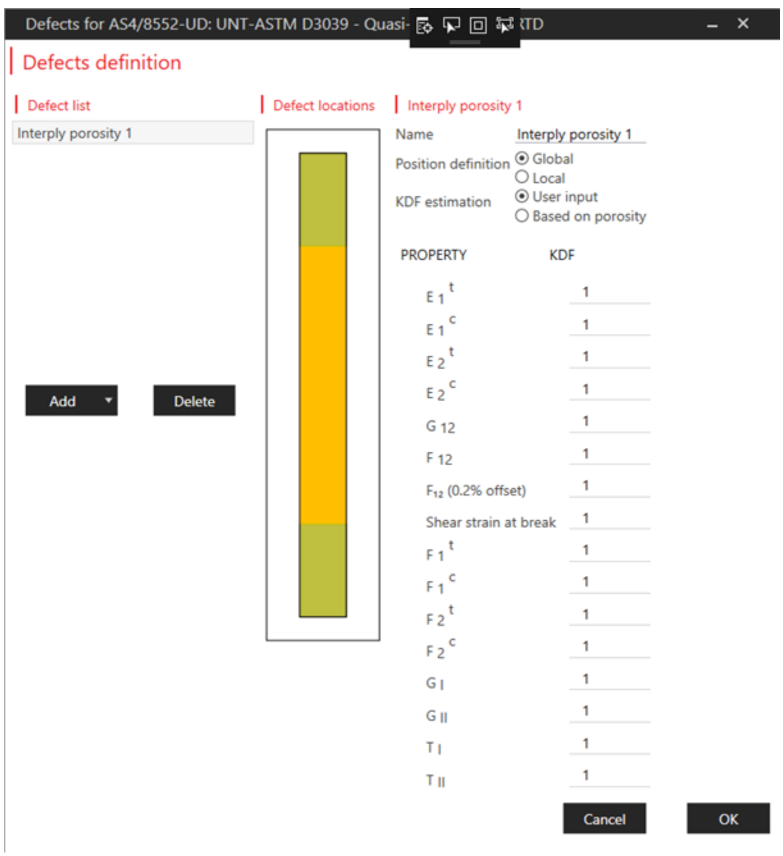


Figure 3-32 Definition of an interply porosity defect using custom KDF.

Initial Delamination

This type of defect obviously requires interface delamination is modelled on the coupon. It is then possible to define an area and a set of interfaces where initial delamination is introduced (see Figure 3-32). At the level of the FEA model, this is achieved by simply removing some of the cohesive elements that are used to model the ply interfaces.

The exact elements that are removed are picked randomly, always ensuring that the constraint on the maximum area of any individual delaminated zone is satisfied (an individual porosity is a set of cohesive elements connected by an edge). After the FEA jobs have been generated, it is possible to inspect for each interface which elements were removed and what is the achieved surface fraction of initial delamination, and to compare that to the requested fraction (see Figure 3-34).

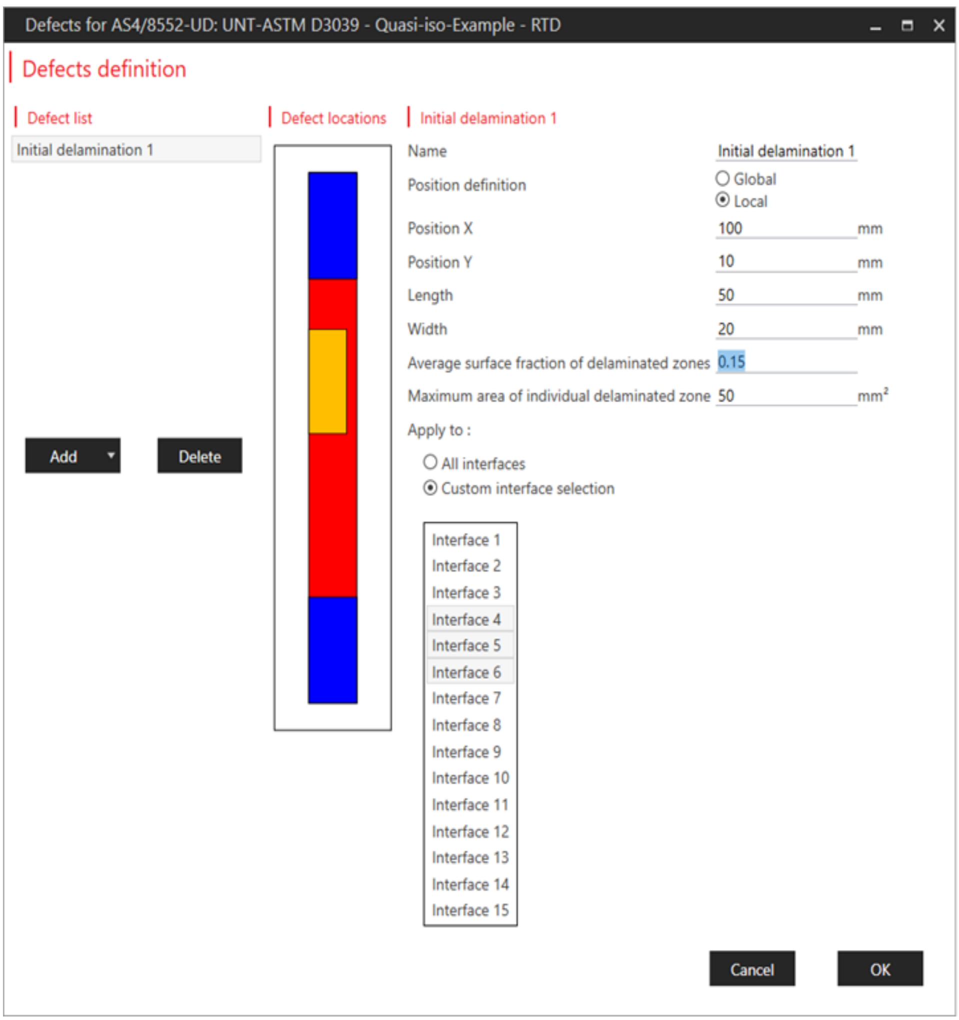


Figure 3-33 Definition of initial delamination defect.

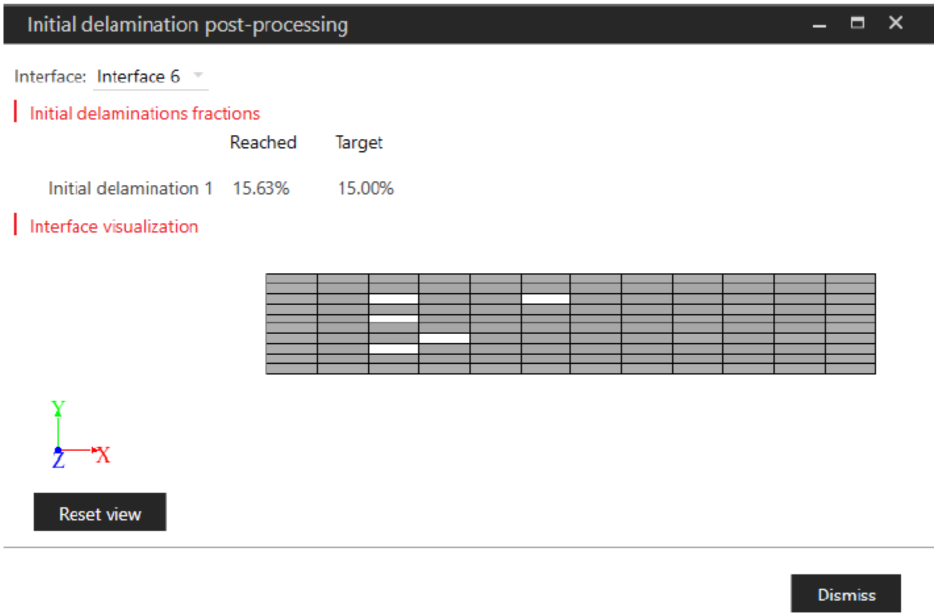


Figure 3-34 Post-processing of the initial delamination defect.

AFP Gap

To consider the effect of the gap induced by automatic fiber placement (AFP), the coupon is modeled via varying the FVF near the gap. AFP gap is defined by the width of the course (number of tow per course *tow width) and the width of the gap. The position of the defect can be defined as global or local, but will always cover the whole coupon area.

- With global position, the defect will be assigned in all the plies and a gap will be always through the center of the coupon. The position of other gaps can be computed based on the width of the course/tow and ply angle.
- With local position, the defect should be defined on specific plies. Gap width and fiber orientation can be varied in each ply. The position of the gaps will be defined by transverse shift, which can be computed by the distance between the coupon center and the nearest gap.

Note:	Only one AFP gap defect can be defined on a coupon.

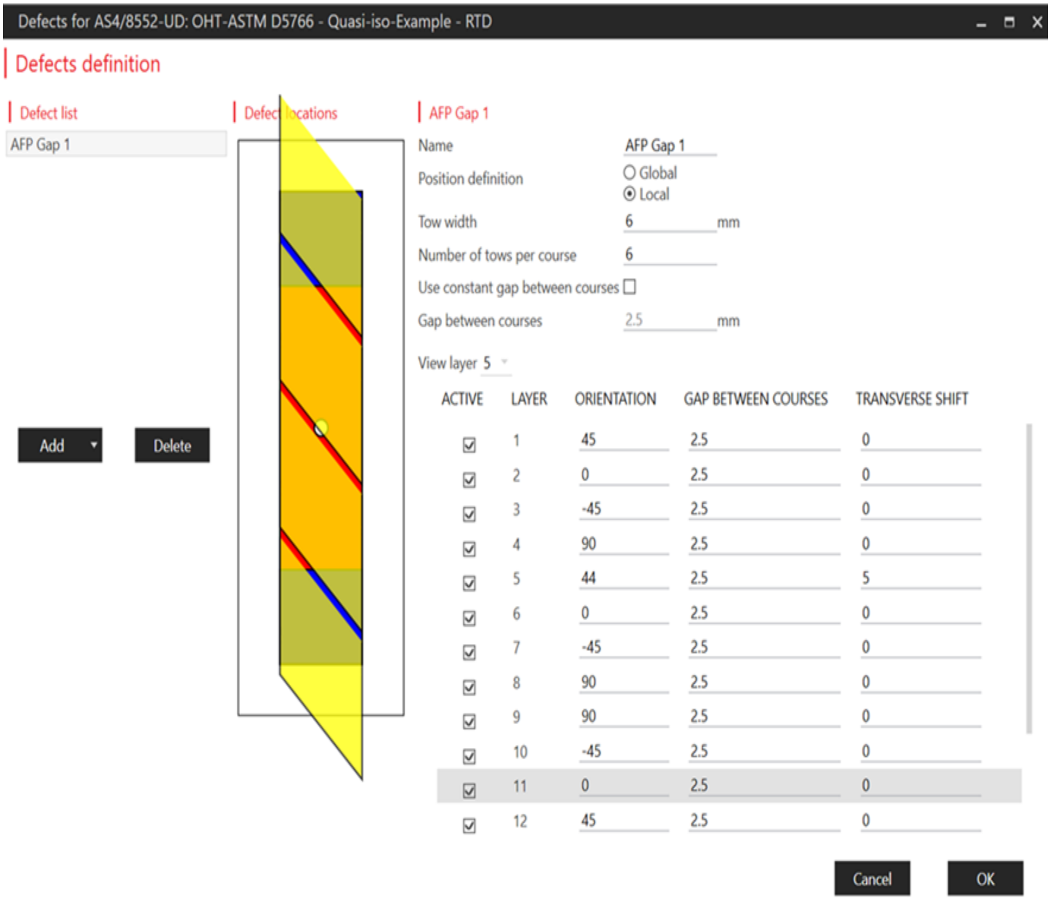


Figure 3-35 Definition of AFP gap defect.

Defect Sensitivity Study

Parametric study can be coupled with the defect study on the defect parameters, which can be activated by opening sensitivity study in variability definition when defect study used (Figure 3-36).

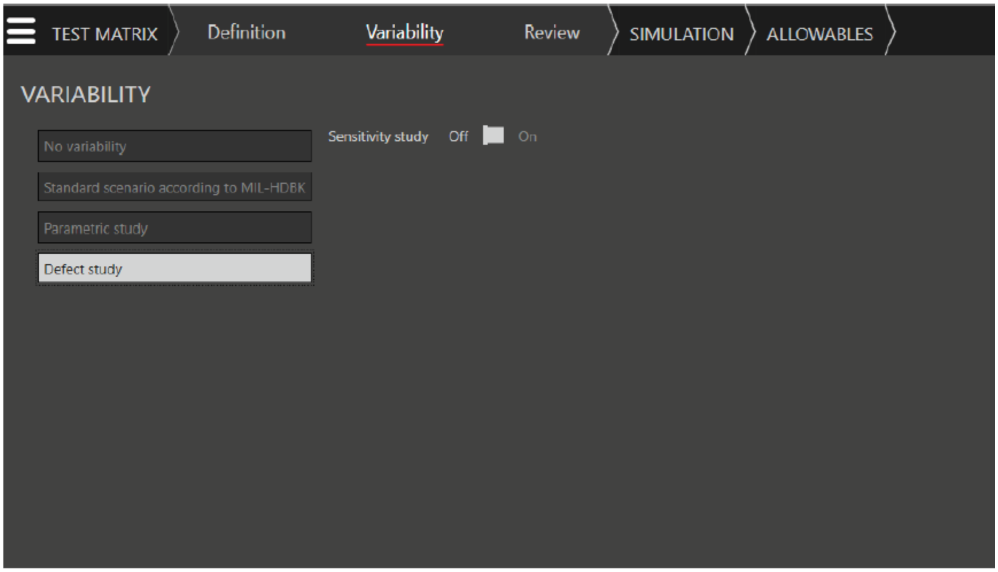


Figure 3-36 Selection of defect sensitivity study.

A window for parametric definition can be opened by clicking the distribution icon when the defect defined in the coupon (Figure 3-37). Each parameter in defect definition can be defined as a variable with a specific distribution.

For continuous variable, it can be defined with random uniform distribution, random normal distribution and user defined distribution. For discrete variable, only random uniform and user defined available.

- For random uniform distribution, the variation should be defined as max value – min value
- For random normal distribution, the variation should be defined as standard deviation
- User defined distribution can be used to define a non-random distribution with 2 available format:
  - A semicolon separated list of discrete values as V1; V2; V3;. . .
  - A range with min value, max value and increment as Vmin : Vmax : inc

3 kinds of sample matrix strategies are provided: full cross matrix, single parameter matrix and Monte Carlo matrix.

- The strategy in full cross matrix and single parameter matrix is similar with the definition in the general parametric study (Parametric Study Definition).
- With Monte Carlo matrix, a completely random sample matrix can be built with the given sample number. For each sample, all the parameters will be randomly picked from the defined distribution.

When a sensitivity study is defined for waviness defects, 2 constraints can be added into the sample matrix for avoiding some unexpected samples:

- Severity range: defined as a ratio between wavelength and max amplitude, which can avoid some terrible wrinkle in layers.
- Fiber content range: the fiber fraction will be varied in waviness domain. The default range in VA is 0
    - 100%. A smaller range on fiber fraction can make the samples more reasonable.

For all kinds of defect, a default constraint will be always enforce that the defect domain must be inside of the coupon. If any constraint is not matched in the sample, the sample will be cancelled and a new sample will be generated to replace it.

Remarks:

- If defining a parametric study on intraply porosity or interply porosity and the porosity is defined as a variable, the material behavior with each porosity value will be estimated with a series of RVE analysis, which may take expensive computation time.
- In current version of Digimat-VA, with a defect coupon, the parametric study only can be defined on the defect parameter, not possible on material behavior or microstructure.

Figure 3-37 Definition of defect sensitivity study.

FEA Job Generation

The job generation process is started by clicking on the **Generate jobs** button. All necessary input files for the FE analyses are generated in the project working directory (and its subdirectories). Individual and a global progress bar give indication about the progress of the generation process. This process can be cancelled at any time.

The FEA job generation is performed in parallel, using half of the available CPUs.

If the CLT computations have been activated, an extra button allows to generate only the CLT results, without actually generating the full finite element models. In that case, the next screen (JOB SUBMISSION) can be skipped to go directly to the GLOBAL POST-PROCESSING screen to view and analyze all the CLT results.

Check Random Variable Draws

Once all FE jobs have been generated, if variability is present in the analysis, it is possible to check all the random variables that have been drawn. This can be performed in the Random variable generation window (see Figure 3-38) which can be opened by clicking on the **normal distribution** icon in the test matrices (see Figure 3-15).

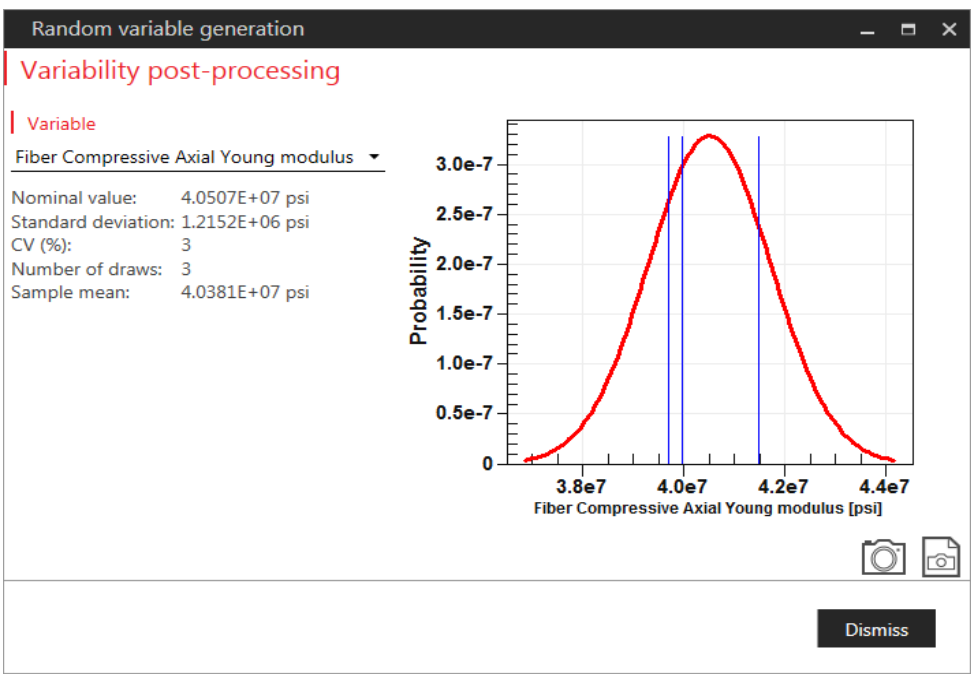


Figure 3-38 Random variable generation window.

Project Unit System

It is important to keep in mind that all FE analyses are created in the project unit system (specified in the startup dialog box or through the Digimat-VA menu). All user input will thus be converted to that unit system prior to the FE analyses generation and the results of the FEA analyses will therefore also be expressed in that unit system. It is however possible to convert the computed allowables to other unit system once all FE analyses have completed.

FEA Job Submission

This screen (see Figure 3-40) allows to monitor FE analyses while they are running and pause or stop the submission of analyses.

The **Submit** button at the bottom left of the screen opens up the **Job submission** window (see Figure 3-41). Two options are available for job submission: local or remote.

When running a test campaign with multiple simulations, it is possible to terminate all running jobs by clicking on the stop button, or alternately, individual simulations may be terminated by right-clicking on the symbol associated to a specific job and selecting the terminate job option as shown in Figure 3-39.

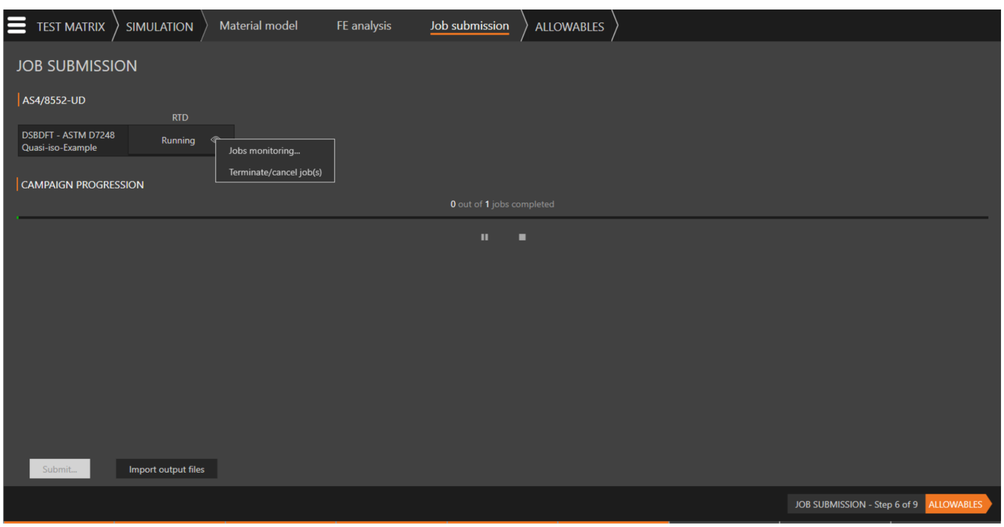


Figure 3-39 Job termination

Local FEA Job Submission

The local job submission is performed on the first tab of the submission window (see Figure 3-41). In this tab, the user can choose the working directory, the total number of processors which can be used simultaneously and the number of processors to use per simulation. The table in the right part of the window allows to define the job prioritization by dragging lines in this table one can change the sequence in which the jobs will run.

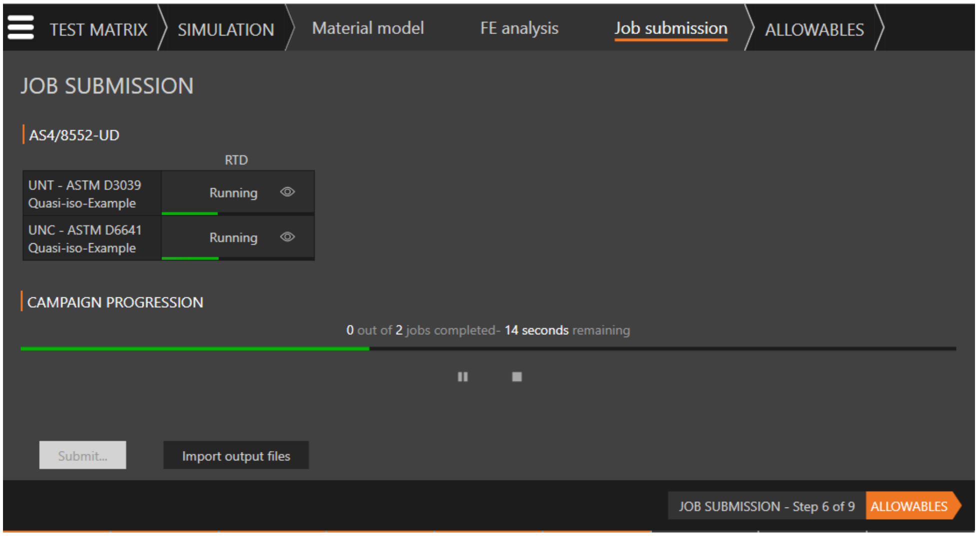


Figure 3-40 Job submission and monitoring.

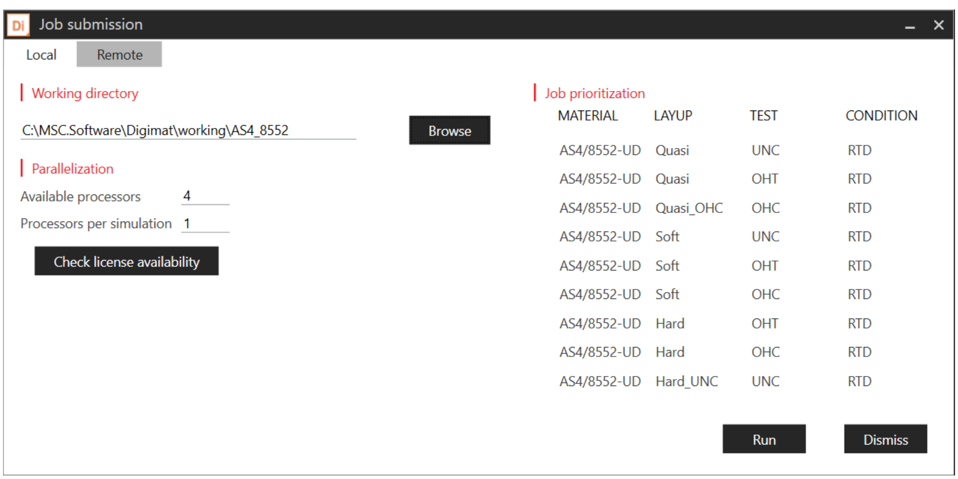


Figure 3-41 Local Job submission window.

Remote FEA Job Submission

The remote job submission is performed on the second tab of the submission window (see Figure 3-42). Remote job submission allows to interactively setup a Digimat-VA test campaign on one computer and running the CPU intensive task (solving all the finite element models) on a different computer (or cluster).

Each individual simulation can be run in parallel on multiple processors by specifying the number of processors per simulation under Parallelization. These parallel simulations will run in shared memory. All processors used for a given simulation must therefore belong to the same node which will be enforced by the submission script. Nevertheless, it is the user responsibility to make sure that the nodes on the remote machine have enough processors to provide the requested number of processors per simulation.

Remote job submission requires having a Digimat-VA installation (same version) on two different machines, one Windows machine for running the graphical user interface and (at least) one Linux machine for running the finite element models. On the Linux installation, the graphical user interface is not available, only the Digimat-VA finite element solver and a small utility in charge of submitting and performing basic job post-processing are available.

The communication between those two machines is done through a SSH connection initiated by the Digimat-VA GUI on the Windows machine. It is possible to define (and store in a persistent way) different remote hosts in the job submission window. A remote host is defined by

- Host name: name or IP address of the remote host
- Port: port to use for the SSH connection
- Authentication method: authentication method for the SSH connection
    - password: a password prompt will popup when the connection is established
    - private key file: provide the path to the file containing the private key to use
- Remote working directory: path to the working directory to use on the remote machine
- Remote Digimat installation directory: path the installation directory of Digimat on the remote machine.

The **Test connection** button will try to connect using the specified information and will check the existence of the remote Digimat working directory and installation directory. When the remote campaign is submitted, the Digimat-VA on the Windows machine will take care of

- Packaging all the necessary input files in a zip archive
- Uploading this archive to the selected remote host
- Submitting the test campaign on the remote host
- Interactively monitor the progress of the test campaign
- Download the results when they become available.

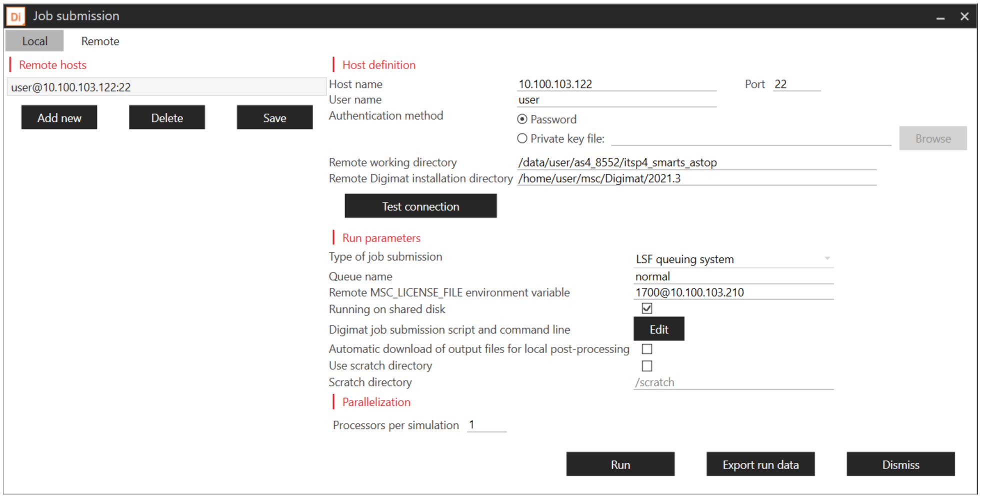


Figure 3-42 Remote job submission window.

Once the campaign has been submitted to the remote host, interactive job monitoring is available on the Digimat-VA Windows in nearly the same way as for a local job submission. The only capability that is not available is the detailed real-time monitoring of a FEA job status file. At this stage, it is possible to save the current project and close Digimat-VA on the Windows machine, the FEA jobs will keep on running on the remote host.

When Digimat-VA is reopened and the project is loaded, Digimat-VA will automatically prompt for reconnection to the selected remote host and enter job monitoring if jobs are still running on the remote host. If all jobs have completed on the remote host, Digimat-VA will automatically download the campaign results. For this capability to work correctly, it is necessary to save the project AFTER the campaign has been submitted (so that the job submission informations can be stored in the project file).

Manual submission on the remote host

For situations where a direct SSH connection between the Windows and the Linux machines is not possible, a manual workflow is available. Using the **Export run data** button in the remote job submission window, Digimat-VA will only perform the first step, packaging all the necessary files in a single Zip archive. It is then the user responsibility to:

- Transfer the archive to the remote host
- Submit the test campaign on the remote host. The command line to use for campaign submission is `feaRemote.sh myZipArchive.zip VA`

  where the `feaRemote.sh` is located in the Digimat installation directory on the Linux machine.
- Transfer the results back on the Windows machine for post-processing, after the campaign completes,

The progress of the campaign can be monitored by looking at the log file generated by `feaRemote.sh`. Once the campaign completes, all the results will be gathered in two zip archives on the Linux remote hosts. One archive contains only **basic** results (the status of each job and the force displacement curve) and the second archive contains the full set of results (i.e. for detailed local post-processing).

Those two archives can be loaded in Digimat-VA on the Windows machine by clicking the **Import output files** button and selecting either one of those two archives. All the available results are then imported in Digimat-VA, the allowables are computed, and usual post-processing operations are available in the exact same way as for a local run or interactive remote run.

Remote FEA Submission Types

There are two main different types of remote job submission: direct or through a queuing system (PBS or LSF).

Direct remote submission

The direct remote works very similarly to the local submission. The only parameter that is necessary is the number of concurrent jobs allowed on the remote machine, and the address of the license server or license file to use when running on the remote machine.

Remote submission to a queuing system

Two different queuing systems are supported by Digimat-VA:

- PBS
- LSF

The workflow is identical for both systems. The following parameters need to be defined (see Figure 3-43)

- **Queue name**
- **Remote MSC LICENSE FILE environment variable:** indicates which license server or license file to use when running on the remote machine.
- **Running on shared disk:** usually only applicable to cluster, it indicates if the hard disk on which the FEA simulations will effectively run can be accessed from the master node (i.e. the one from which the jobs are submitted). This is necessary to have a continuous monitoring of the job progress. If the jobs are not running on a shared disk (which is the case for instance if the queuing system takes care of transferring the job input files from the master node to the compute node and back to the master node when the job completes) the progress will not be updated when the job is running. It will only be available when the job completes and the job output files are transfered back onto the master node.
- **Job submission script and command line:** allows to specify the job submission script and the command line (and command line options to use)

- **Automatic download of output files for local post-processing.** When this option is checked, the full set of output files is automatically downloaded when the campaigns completes on the remote host.

Submission Script for Queuing System Submission

The submission of a job to a queuing system is usually performed through a script that allows to define the necessary parameters, environment variables,... A basic default script example is provided for PBS and for LSF queuing system. Because of the very large number of jobs to be submitted, what is defined in Digimat-VA is only a *master* script, which can use different variable (keywords enclosed in percentage signs).

This master script will then be customized by Digimat-VA on the remote host for each and every FEA job to submit all the keywords enclosed between percentage signs will be substituted by their value at this stage. The list of available variables is shown in the submission script dialog box (see Figure 3-44).

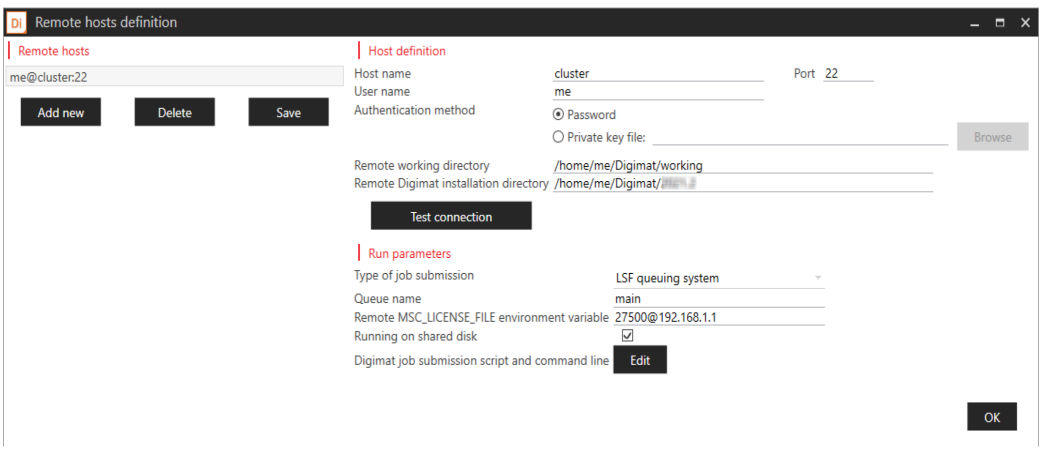


Figure 3-43 Remote job submission window for submission to a queuing system.

Note that the provided default script is tailored for Digimat-VA 2023.3. Reusing the same script on another version of Digimat may require some small modifications. When importing a script through the **Import script** button, some common incompatibility errors are automatically detected and reported.

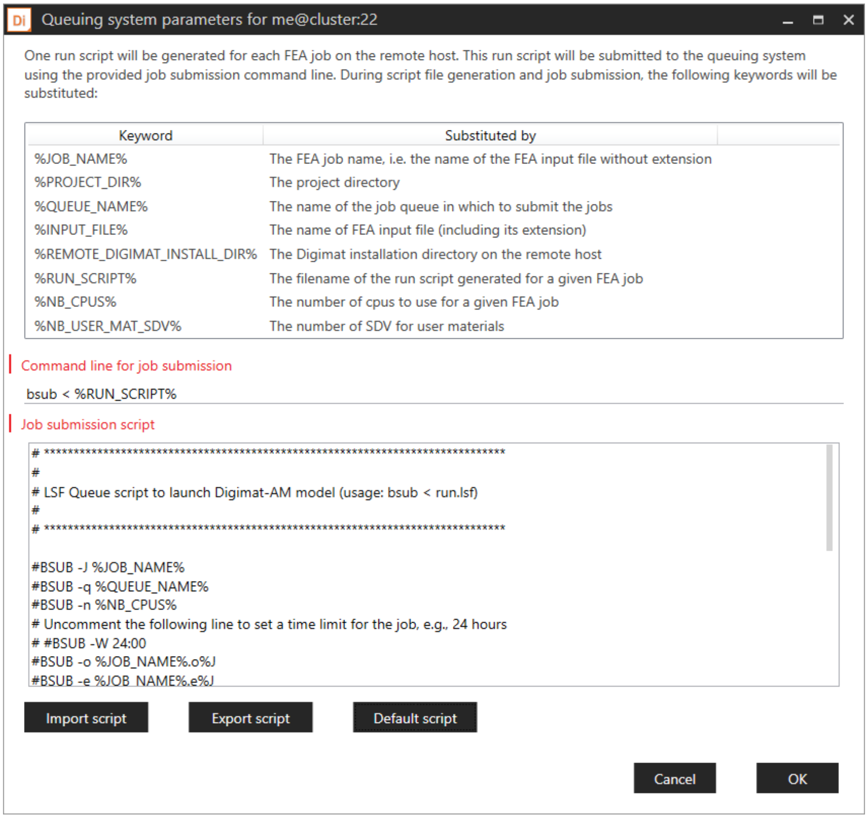


Figure 3-44 Queuing script for remote job submission to a queuing system.

FEA Job Monitoring

Once jobs are running for one (or more) virtual tests, it is possible to access a more detailed monitoring by clicking on the eye icons in the table. It opens up the window in Figure 3-45 in which it is possible to consult the detailed status of a specific FE analysis.

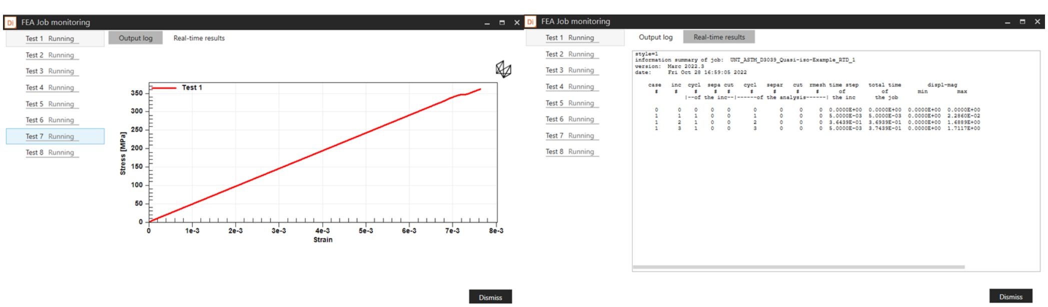


Figure 3-45 Detailed job monitoring (shows all the FE analysis for one specific virtual test).

In certain cases, the results curves will not be updated till the simulation has successfully completed. For these specific cases, the selected graph will remain empty till the results are available. These specific cases include:

- Fracture resistance curves (R-Curves) for DCB, C-ELS and ENF tests
- Energy balance curves for all impact simulations.

4 Allowables

- Global Post-processing
- Computation of Allowables
- Local Postprocessing

Global Post-processing

The global post-processing screen allows to see the values of the computed allowables for each entry of the test matrix, as shown in Figure 4-1. These values are automatically refreshed each time a job finishes.

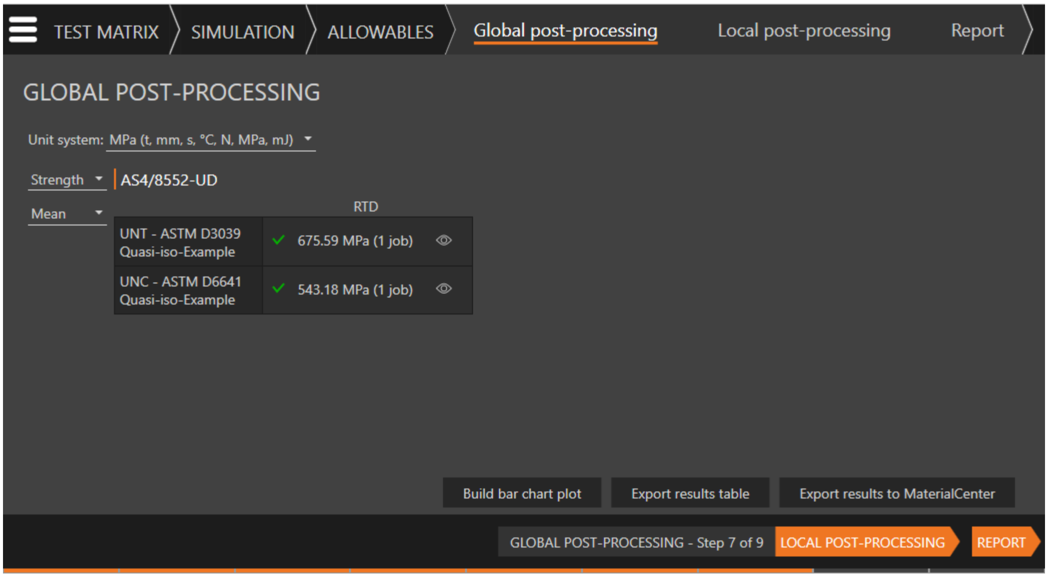


Figure 4-1 Values of the computed allowables.

Depending on the workflow chosen at the Variability substep in the Test Matrix step, the available detailed post-processing may vary:

- The **No variability** workflow provides mean allowable values for stiffness, strength, and flexural modulus where applicable, as well as raw stress-strain curves except in the case of drop weight impact tests.
- The **No variability** workflow in the case of drop weight impact and tension/compression after impact tests, provides mean allowable values for residual dent depth, absorbed energy, damaged area and delamination area as well as a force-displacement curve, a force evolution curve, a velocity evolution curve, an absorbed energy evolution curve and an energy balance. In addition, for the tension/compression after impact test, some results are shown separately for the impact step and for the compression step.
- The **No variability** workflow in the case of the double cantilever beam test results in the following deliverables being generated which are computed based on the standard ASTM D5528:
    - Mode 1 Toughness (MCC): Modified Compliance Calibration Method
    - Mode 1 Toughness (CC): Compliance Calibration Method

  - Mode 1 Toughness (MBT): Modified Beam Theory Method
- The No variability workflow in the case of the calibrated end loaded split test results in the following deliverables being generated which are computed based on the standard ISO 15114:
  - Mode 2 Toughness (ECM): Experimental Compliance Method
  - Mode 2 Toughness (SBT): Simple Beam Theory
  - Mode 2 Toughness (CBTE): Corrected Beam Theory using Effective Crack Length
- The No variability workflow in the case of the end notched flexure test generates the following mean allowables in accordance with the norm ASTM D7905:
  - Mode 2 Toughness (CC): Compliance Calibration Method
- The No variability workflow in the case of the bearing tests for the norm ASTM D5961 generates the following additional mean allowables:
  - Offset strength
- The No variability workflow in the case of the bearing tests for the norm ASTM D7248 generates the following additional mean allowables:
  - Offset strength
  - Gross bypass strength
  - Net bypass strength
- The **Standard scenario according to MIL-HDBK** workflow provides on top the A and B-basis for strength.
- The **Parametric study** workflow provides mean values for stiffness and strength, as well as most importantly the raw results and 2D plots that highlight the influence of parameters on stiffness and strength.

Bending tests additionally provide the option to switch between stress-strain curves and force-displacement curves (see Figure 4-2).

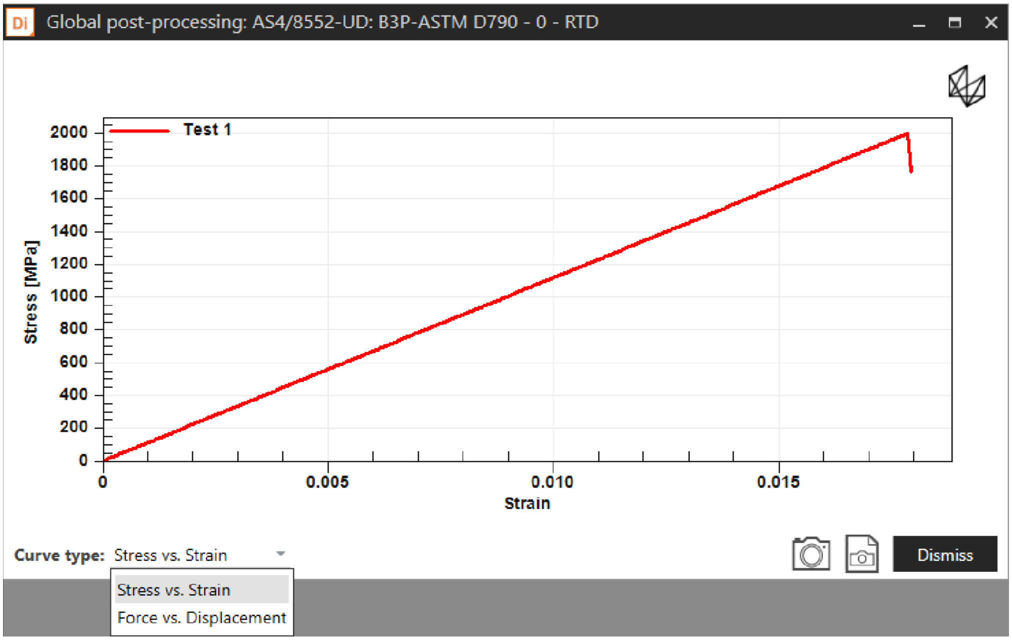


Figure 4-2 Global post-processing window showing the possibility to switch between stress-strain or force-displacement curves.

For the DCB and C-ELS tests, the following curves are generated:

- Force vs. Opening Displacement
- Crack Length vs. Time
- Fracture Toughness vs. Crack Length

For bearing tests, the following curves are generated:

- Stress vs. Strain
- Force vs. Displacement
- Bypass Stress vs. Displacement

Detailed View for Variability and Standard Scenario According to MIL-HDBK

Click on the eye icon associated to an entry of the test matrix to open a window (see Figure 4-3) containing more details about the jobs related to that entry.

First, the exact strength/stiffness value obtained from each job is shown in the histogram at the left, while the values of the allowables are displayed at the right. Clicking on a bar in the histogram brings up the variability diagram at the bottom right. This diagram shows, for each variable parameter, the random draws used for all jobs (in blue), and the particular draw used for the selected job (in red). The exact numerical values are also displayed next to the diagram.

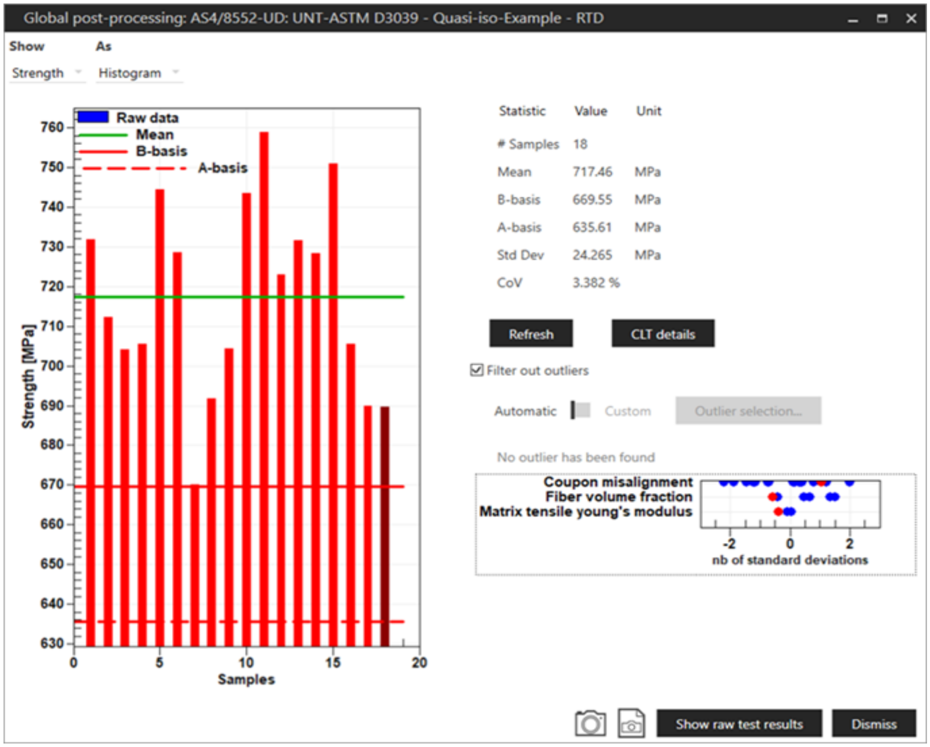


Figure 4-3 Global post-processing window showing the global histogram and the variability diagram.

It is also possible to display a distribution function plot instead of a histogram, as shown in Figure 4-4. This plot allows you to easily see the proportion of samples with a strength/stiffness value lying in a given interval.

It is also possible to display the raw stress-strain curves related to the jobs, from which the strength/stiffness values are extracted. Click **Show raw test results** to bring up a window containing the stress-strain curves for all jobs of the current entry of the test matrix (see Figure 4-5).

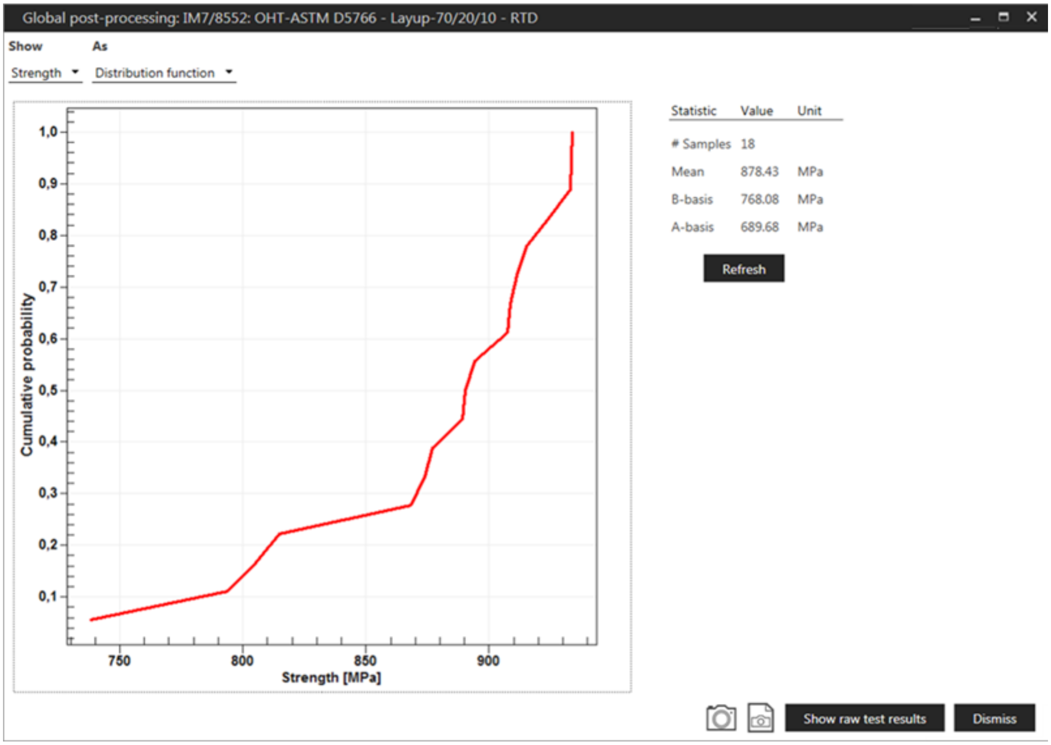


Figure 4-4 Global post-processing window showing the distribution function plot.

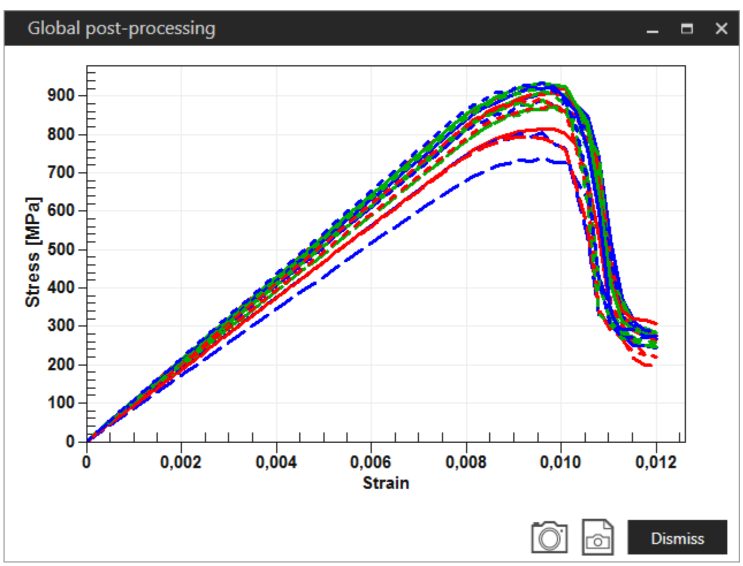


Figure 4-5 Raw stress-strain curves.

Comparison: Allowables Values Across the Whole Test Matrix

In the main global post-processing screen, clicking the **Build bar chart plot** shows a visual comparison of allowables values between all entries of the test matrix (see Figure 4-6).

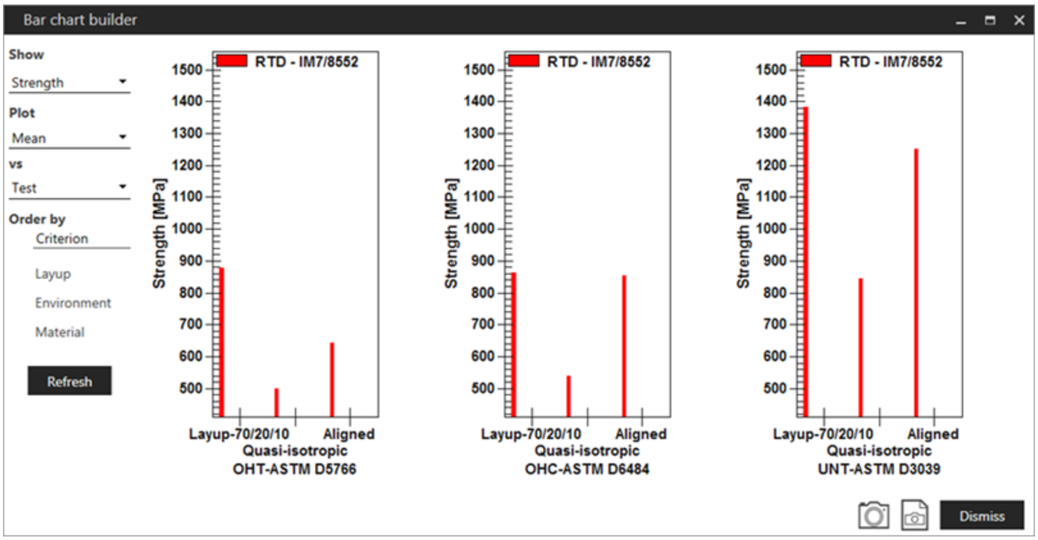


Figure 4-6 Comparison of allowable values using bar chart plots.

In this window, the values of the selected allowable (e.g., strength B-basis) is shown for all materials, all layups, all tests and all environments. The results are displayed as a collection of plots, depending on the selected criterion (here, one plot per test). Inside each plot, the values of the allowables are displayed as histograms, grouped according to the order shown in the left part. This order can be modified by using a drag-and-drop gesture between the three available criteria.

View of Parametric Study

Click on the eye icon to open the **Parametric study post-processing** window, which consists of two tabs. In the first tab the raw results can be visualized one-by-one, with the corresponding varied parameter values and the stress-strain curve (refer Figure 4-7). Each result is linked to a job id that may be useful later on in the local post-processing.

By activating the **show relative output values**, you can quickly obtain an idea of the relative sensitivity of each parameter. By switching to the second tab a 2D plot can be build for stiffness or strength against the varying parameters (refer Figure 4-8). Each axis may be expressed in relative terms to ease the understanding of the sensitivity.

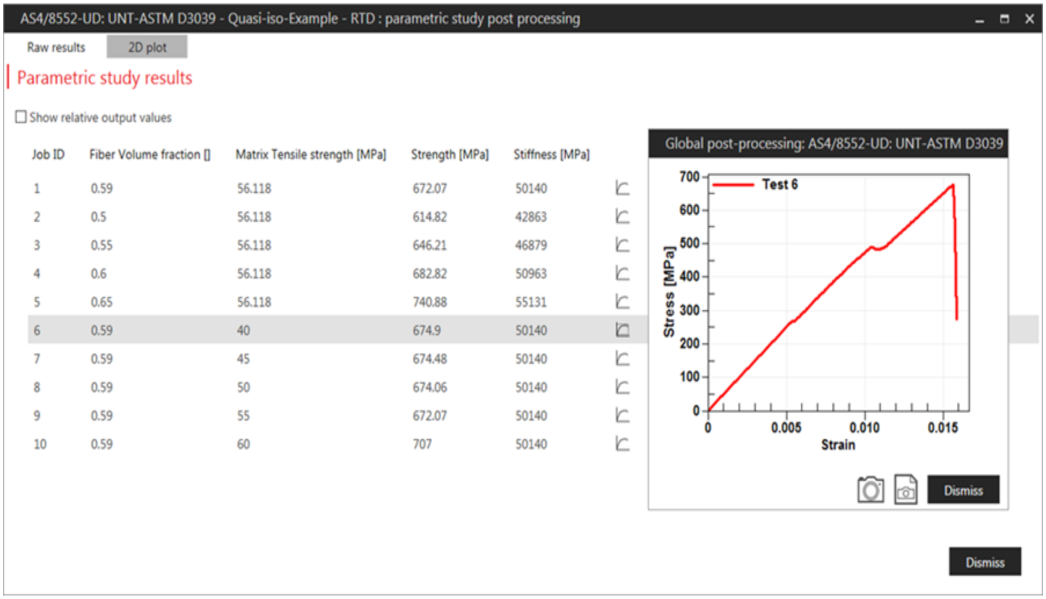


Figure 4-7 Visualization of raw parametric study results.

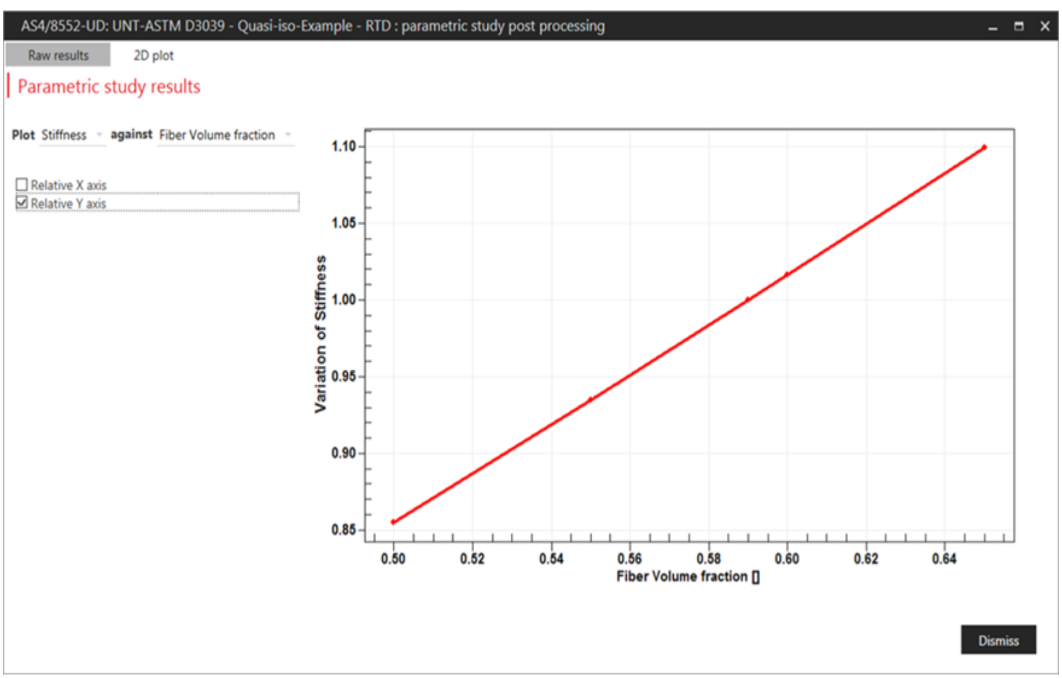


Figure 4-8 2D plot of parameter influence on coupon stiffness or strength.

Detailed View for Defect Sensitivity Study

In global result, the statistics result (mean/A-basis/B-basis) can be provided based on the sample matrix. Clicking the eye icon, a post processing window can be opened for defect parametric study. As for the general parametric study, raw result and 2D plots are provided in the first and second tabs in the window. In the third tab, the statistic result on each variable can be provided with linear or quadratic regression. Fitting function, dispersion and weight can be given.

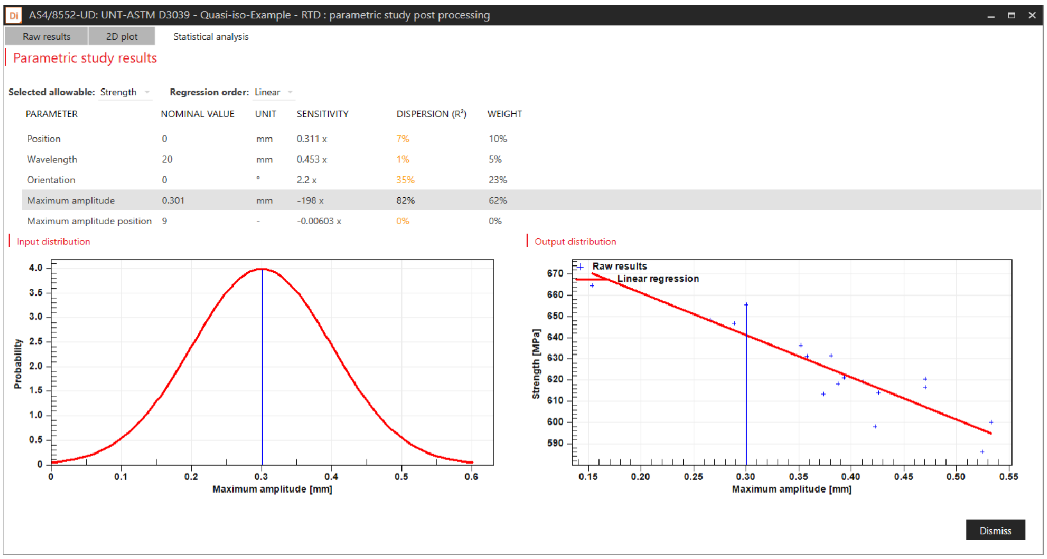


Figure 4-9 Post processing window for a defect parametric study.

Results Export

Click on **Export result table** to generate a Microsoft Excel spreadsheet (`.xlsx` file) containing allowable values (mean strength, strength B-basis and A-basis, mean stiffness), variability random draws and the raw data used to compute the allowable values, i.e., points of the stress-strain curves.

This information is available for all jobs and is organized such that each material has its own dedicated sheet in the document.

Results Export to MaterialCenter

Click **Export to MaterialCenter** to generate a dedicated Microsoft Excel spreadsheet (`.xlsx` file) per material system, containing the raw results. Each spreadsheet is named through the concatenation of the defined export file name and the corresponding material system name.

Each spreadsheet can be then be imported in MaterialCenter in order to store and manage the virtual data generated by Digimat-VA at the enterprise level. Further details on the usage of the Digimat-VA interface to MaterialCenter are available in the dedicated section Interface to MaterialCenter.

Carpet Plot

If carpet plot layups were added to the test matrix, the eye icon of the corresponding test matrix entry will bring up the carpet plot window, as shown in Figure 4-10.

In this plot, each data point (corresponding to a specific layup) can be clicked to see the detailed global post-processing for that specific layup (like Computation of Allowables).

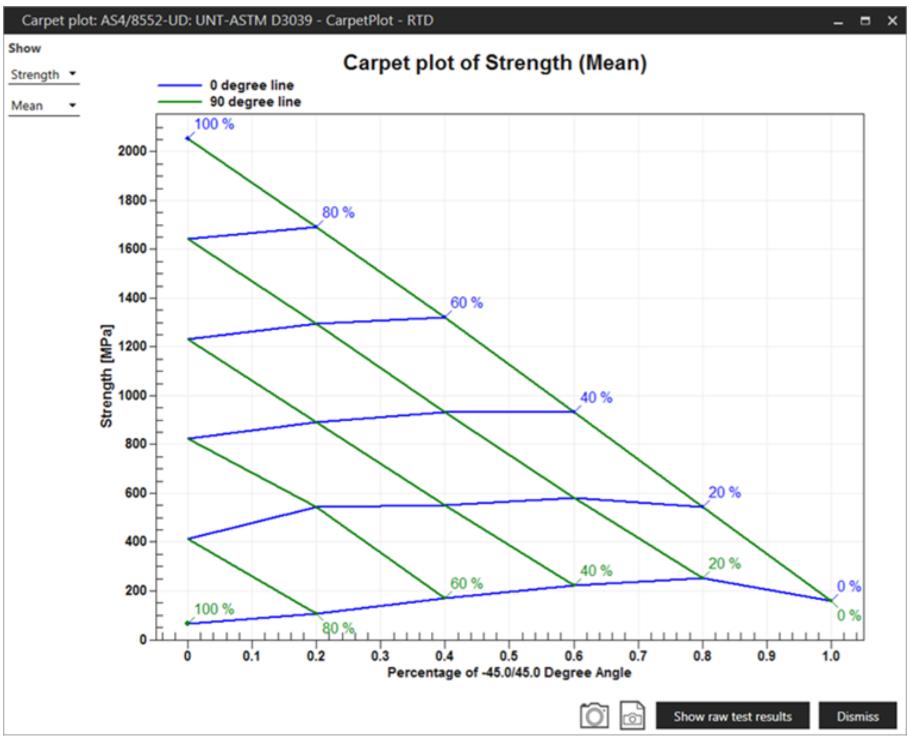


Figure 4-10 Carpet plot.

Computation of Allowables

In Digimat-VA, when using the **Standard scenario according to MIL-HDBK** variability workflow, the values of the allowables are automatically computed from the raw strength/stiffness values extracted from the stress-strain curves. This section documents the process used to compute these allowables. Digimat-VA follows the methodology from the CMH-17.

Two allowables are computed in Digimat-VA:

- The B-basis, which represents a 95% confidence lower bound on the tenth percentile of the population.
- The A-basis, which represents a 95% confidence lower bound on the first percentile of the population.

These values are computed as follows. Let $x_1, \ldots x_n$ be the raw data values. The sample mean $\bar{x}$ and sample variance s_x^2 are defined to be:

$$\bar{x} = \frac{1}{n}\sum_{i=1}^{n} x_i, \; s_x^2 = \frac{1}{n-1}\sum_{i=1}^{n}(x_i - \bar{x})^2 \tag{4-1}$$

Provided that the number of samples *n* is at least 4, the B-basis is defined as $\bar{x} - k_B s_x$ where

$$k_B = 1.282 + \exp\left(0.958 - 0.520 \ln n + \frac{3.19}{n}\right) \tag{4-2}$$

The A-basis is defined as $\bar{x} - k_A s_x$ where

$$k_A = 2.326 + \exp\left(1.34 - 0.522 \ln n + \frac{3.87}{n}\right) \tag{4-3}$$

Two options affecting the computation of the allowables are provided

- Normalization
- Outlier filtering

Allowable Data Normalization

In samples where the fiber volume fraction is allowed to vary, it is usually recommended to normalize fiber dominated properties before computing statistics. The objective is that fiber volume fraction should not be included as a source of variability, it is assumed to influence in a linear way the fiber dominated properties.

The normalization method used in Digimat-VA is the following:

```
normalized value = initial value * (FVF specimen / FVF reference)
```

It is only applied to stiffness and strength values.

Outlier Filtering

An outlier is an observation that is much lower or higher than the other observations in the current data set. Outliers usually have to be removed from the data set before computing statistics. The methodology implemented in Digimat-VA for filtering out the outlier is the Maximum Norm Residual methodology (MNR) from the CMH-17. If we assume a statistical population xi, the norm residual is the ratio $|x_i - \bar{x}| / \sigma$ (where $\bar{x}$ is the mean value and σ is the standard deviation).

The norm residual is computed for each sample of the population and compared to the critical values provided by CMH-17 (that are varying as a function of the number of samples). If the value of the norm residual is higher than the critical value, it is considered as an outlier and removed from the population before computation of the allowables.

The outlier filtering can be controlled either globally, at the test matrix level, or locally for each virtual test, in the corresponding **Global post-processing window** (see Figure 4-3). It is also possible to manually indicate one or several test results as outlier and exclude them from the allowable computation. The switch from automatic to manual filtering can only be performed at the virtual test level, in the Global post-processing window (see Figure 4-3). The **Outlier selection...** button then gives access to a new window (see Figure 4-11) showing the value of the norm residual for each sample and a checkbox allowing to select the sample as an outlier.

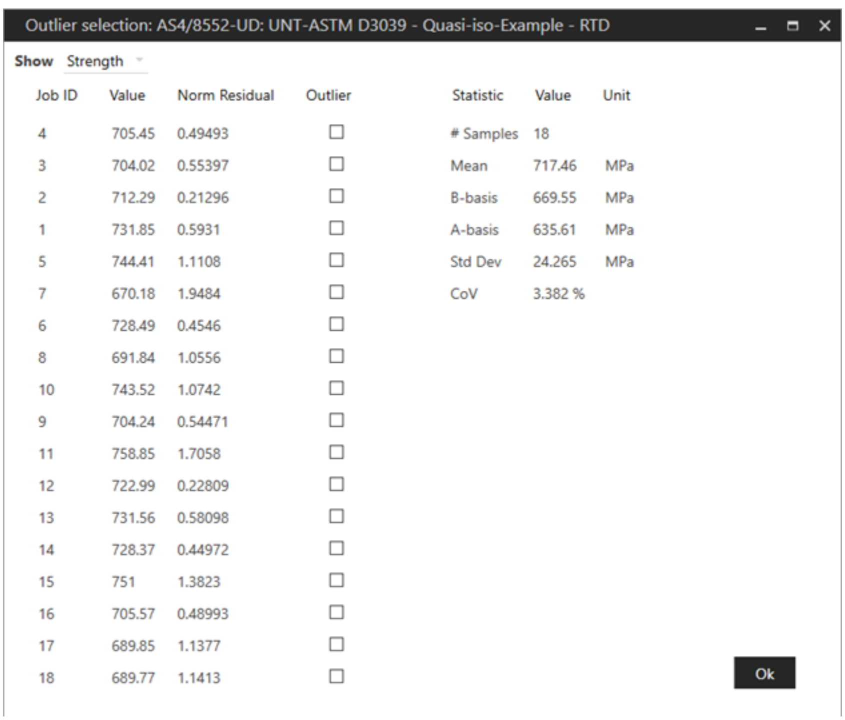


Figure 4-11 Manual outlier selection.

Local Postprocessing

As with the global post-processing screen, the local post-processing screen also allows to see the values of the computed allowables for each entry of the test matrix (see Figure 4-1). These values are automatically refreshed each time a job finishes.

However, the detailed view here is used to check results at a local level on the FE model, for example to visualize the failure pattern.

Detailed View

In the detailed view, the window is divided into six regions:

- In the center, the visualized model itself, as well as some buttons to center or rotate the view.
- At the left, the list of all jobs related to the selected entry of the test matrix is displayed, allowing to choose which one is displayed. These tests are ordered as in the global post-processing histogram in Figure 4-3.
- At the right, the time step for which the results are shown can be selected using the slider. The stress-strain curve of the selected job is also shown, with the red circle corresponding to the value at the selected time step. As a reminder (see *Maximum number of time steps* in Simulation), results are not saved and, hence, cannot be displayed at all increments when using the Advanced PFA model.
- At the top, the user can choose which result to display. Available fields depend on the outputs which have been enabled in the FEA settings used to generate the jobs and of the failure modeling strategy.
- At the bottom, there are some options related to mesh visualization, contour plot smoothening, and model deformation scale factor.
- At the bottom-right of the window, next to the **Dismiss** button, there are three buttons to:
    - Take a high-quality snapshot of the model visualization. Note that this button opens an intermediate dialog which provides more detailed snapshot options (change image quality, save to disk or to clipboard, etc…)
    - Take a snapshot (with standard quality) and add it to the report
    - Create an animation. Note that this button opens an intermediate window which provides detailed animation generation options as follows:
        - Selecting the filename and location of the animation file (`.gif`)
        - Using a transparent background for the animation
        - Resizing the image
        - Modifying the frame rate
        - Creating animation loops

- Modifying the output frequency
- And options to optimize the memory consumption when generating animations.

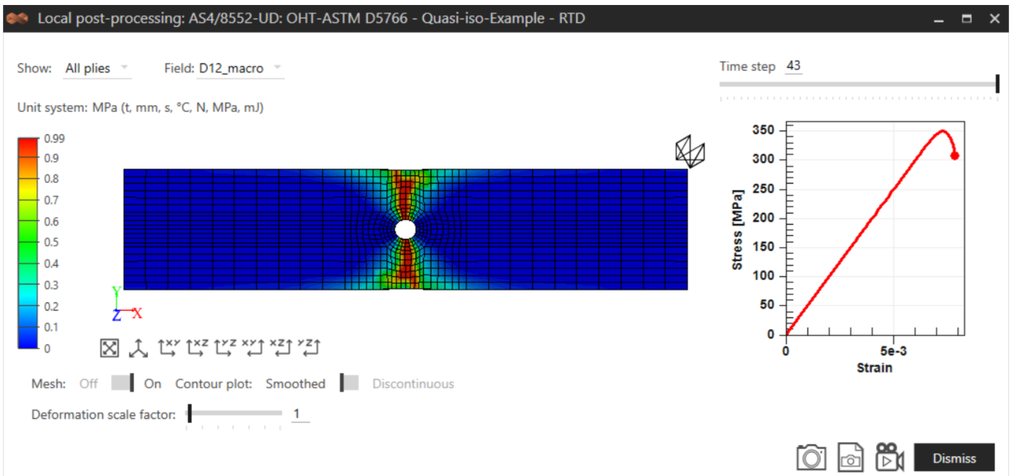


Figure 4-12 Local post-processing window showing the failure pattern.

In the case of bending, short-beam strength, end notched flexure, drop weight impact and tension/compression after impact tests which have been run with the external solver, it is possible to (de)activate the visualization of loading and support structures.

Drop weight impact and tension/compression after impact tests additionally provide the possibility to visualize the delaminated (interface) and damaged (ply) area through the thickness of the laminate. This option is available within the **Field drop down** selection.

Color Scale Manager

The color scale used during post-processing can be fully customized by using the **Color scale manager**. The latter is accessible by clicking on the color scale on the right-hand side of the screen, or through the **General settings**, in the **Visualization** tab.

The **Color scale manager** (see Figure 4-13) offers the possibility to:

- Change the colors used by the color scale (color type and number of colors), including for out-of-range values;
- Use a continuous (Figure 4-14) or discrete (Figure 4-15) color scale;
- Define the minimum and maximum values of the color scale;
- Save a color scale in the manager for later use. The color scale becomes accessible from the drop-down list located on the right;
- Export a color scale to a file (`.dcs` file). Importing `.dcs` files is also possible;

- Save a color scale as default. This scale will be used by default in future sessions of Digimat-VA;
- Change the font size of the color bar. This is useful to make the text more visible for snapshots.

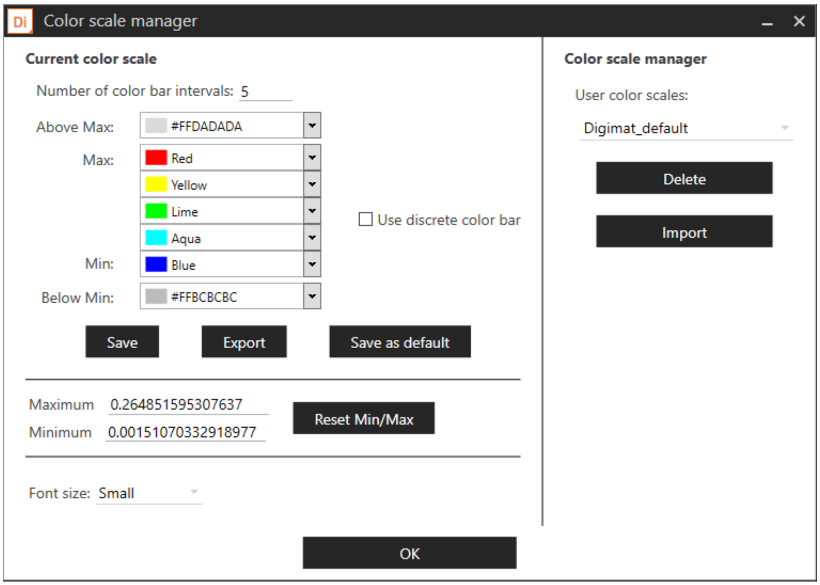


Figure 4-13 Color scale manager.

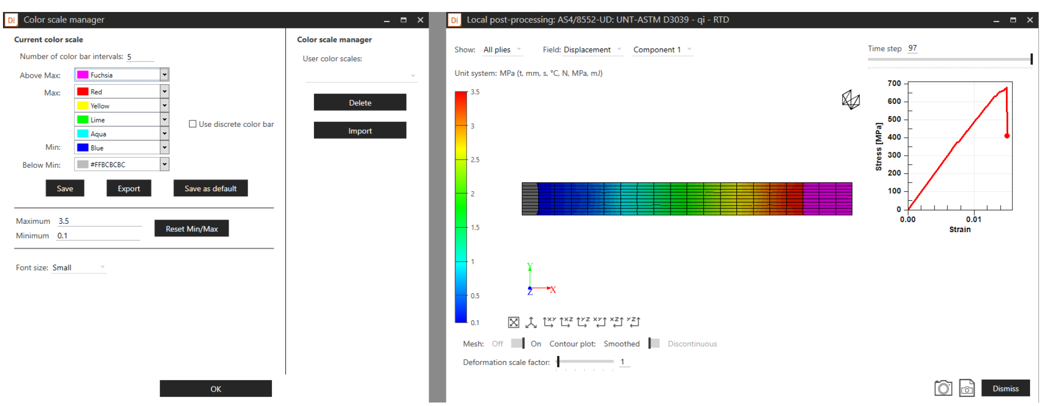


Figure 4-14 Example of a continuous custom color scale.

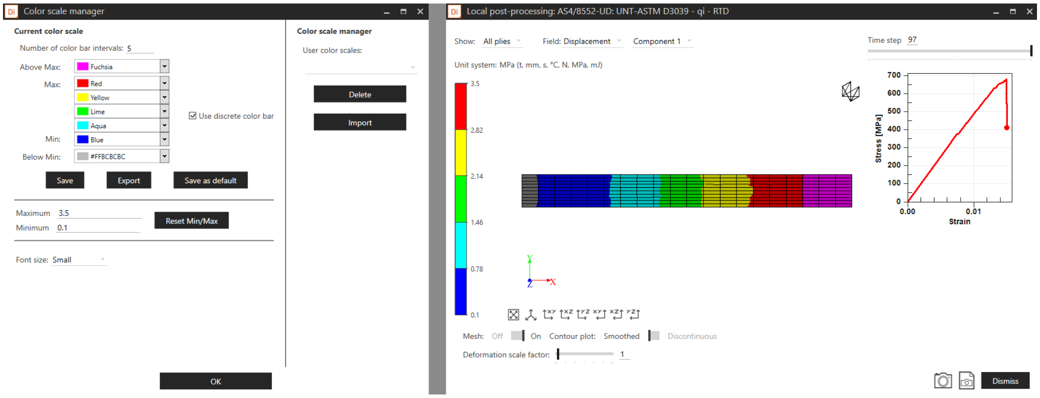


Figure 4-15 Example of a discrete custom color scale.

Report

The report screen (see Figure 4-16) allows to automatically generate a report which contains information about the whole campaign, e.g., the description of all tests, the Digimat models used in the simulation, or the values of the allowables obtained after completion of the jobs. This information is presented as a list of tables and user-created plots.

The report is generated as a Microsoft Word document (`.docx` file), which allows the user to edit it easily or to create his own report from parts of the generated report.

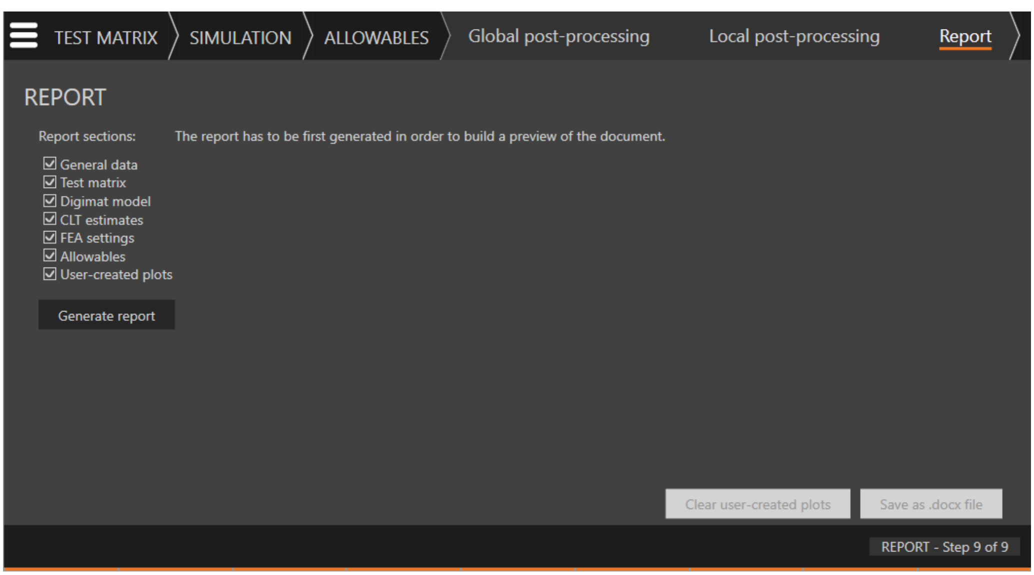


Figure 4-16 Report screen before generation.

Report Generation

The generation of a report is done using the following steps:

- First, the content of the report can be customized using the checkboxes at the left. Only enabled sections will be included in the report.
- Use the **Generate report** button to create the customized report. A preview of Microsoft Word is installed on the computer.

  If the report is generated before the end of the campaign, all currently available results will be included. An example of preview is shown in Figure 4-17. The content of the preview is fully selectable and searchable; print and zoom options are also available.
- If the preview is satisfactory, click the **Save as .docx file** button to save the `.docx` file**.**

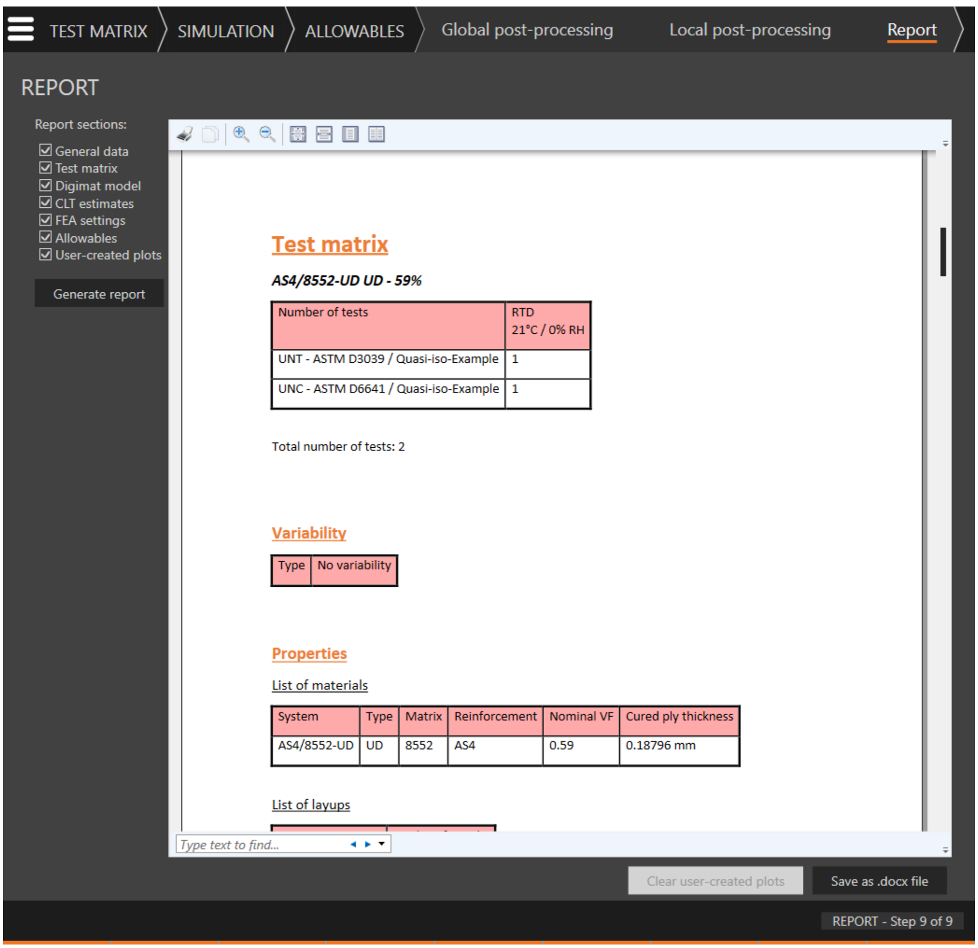
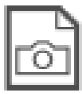


Figure 4-17 Example of report preview.

User-created Plots

A particular section in the report is the user-created plots section. This section contains a collection of figures that have been added to the report by the user.



In Digimat-VA, most plots can be added to the report by clicking on the corresponding (here, **Add plot to report**) button. Each plot will appear in the user-created plots section, in the order of insertion into the report. The section can be cleared using the **Clear user-created plots** button.

5 File Data Management

- Project and Files
- Database
- Interface to MaterialCenter

Project and Files

A project in Digimat-VA can generate large amounts of data. Those data are always created in the project directory, specified at Digimat-VA startup. Inside the project directory, the following directory structure is used:

- One directory for each material system
  - One directory for each association test - layup - environment (corresponding to a virtual test)
    - One directory for each test realization. This directory will contains all files related to one finite element analysis (mesh, Digimat material files, finite element log file and result file).

Project directories are never deleted by Digimat-VA.

A Digimat-VA project can be saved to disk in two different formats:

- The `.vlp` file (Digimat-VA light project): to reduce the project file size, only the results of the virtual tests are saved (i.e. values of strength, stiffness and the stress-strain curves of each test), but the finite element analyses files are not saved (mesh, Digimat material files, results file).
- The `.vcp` file (Digimat-VA complete project): contains everything that is present in the project directory.When this project is loaded back in Digimat-VA, the full project directory is restored to disk.

Both formats always contain the full test matrix definition, variability, Digimat material models and FEA settings.

In the working directory of each test, a file with the extension `.var` is generated and updated while the simulation is running. This file contains the raw curve data which can be seen in the job submission window and varies depending on the type of test which is being simulated.

Database

Digimat-VA uses a database to store all input data in a persistent way. The following data can be stored in the database:

- Material system
- Layups
- Test
- Environment
- Material models (Digimat models associated to material systems, for various environments)
- FEA settings

The database is stored in a single file (using SQLite database format) stored in a dedicated subdirectory of the installation directory of Digimat-VA (in `Digimat/2023.3/DigimatVA/db`). This subdirectory is not deleted by the uninstaller.

Manually deleting this subdirectory or its content will lead to permanent and irretrievable loss of the information stored in the database. It is mandatory to have read/write access to this directory in order to be able to start and use Digimat-VA. In situations where Digimat is installed in a location where users don't have write access, it is possible to move the database to a different directory. In that case, it is necessary to adjust accordingly the value of the `LocalDatabase_SQLite_Directory` key in the `DIGIMAT_Settings.ini` file, which is used by Digimat-VA to locate its database files.

When installing a new version of Digimat-VA, a new version of the database is always created. The older versions of the Digimat-VA database can still be accessed:

- By using the **Tools** menu of the **database manager** window. Click **Change database** entry and select the `vadb.db` file from another previous installation. This change only affects the current session of Digimat-VA.
- By replacing the `vadb.db` file of the current Digimat-VA installation by one from a previous installation. This change will then be permanent.

The database manager window can be accessed through the Digimat-VA menu (see Figure 5-1). It has one tab for each type of data stored in the database.

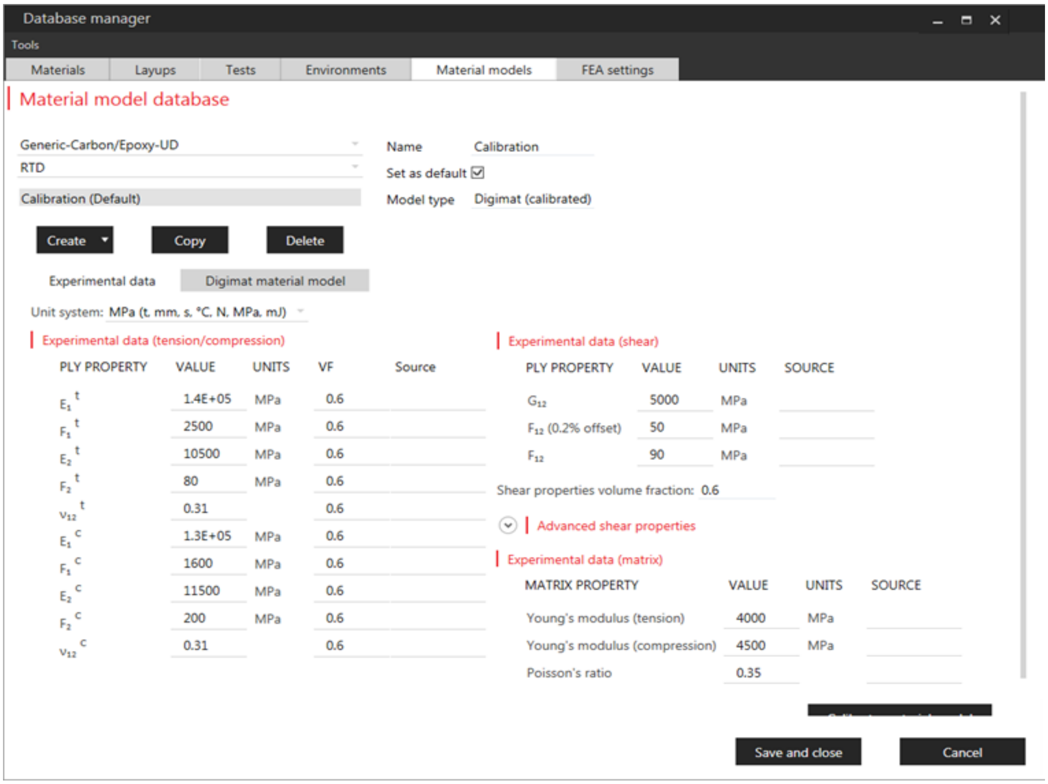


Figure 5-1 Database manager window.

For each type of data, it is possible to edit, create or delete an item (in the same way as items are created in the test matrix definition step). Using the menubar in the **database manager** window, it is possible to create a backup copy of the database file and to select a different database file from which to read data.

Note that all changes performed in the **database manager** window are only committed to the database when clicking the **Save and close** button. Closing the window or clicking the **Cancel** button will cause all changes to be lost.

Material Data Import

The **material models** tab in the database manager shows the different material models available for each system and environment defined in the database (in the upper left area of the tab). The lower part of the tab shows the selected experimental data and/or Digimat model. Using this tab, it is possible to create new material models, either by providing experimental data and calibrating a Digimat model or by directly importing an existing Digimat model.

Import Digimat Material Model

The import **Digimat model** window (see Figure 5-2) allows to provide custom Digimat model to be used for a specific material system.

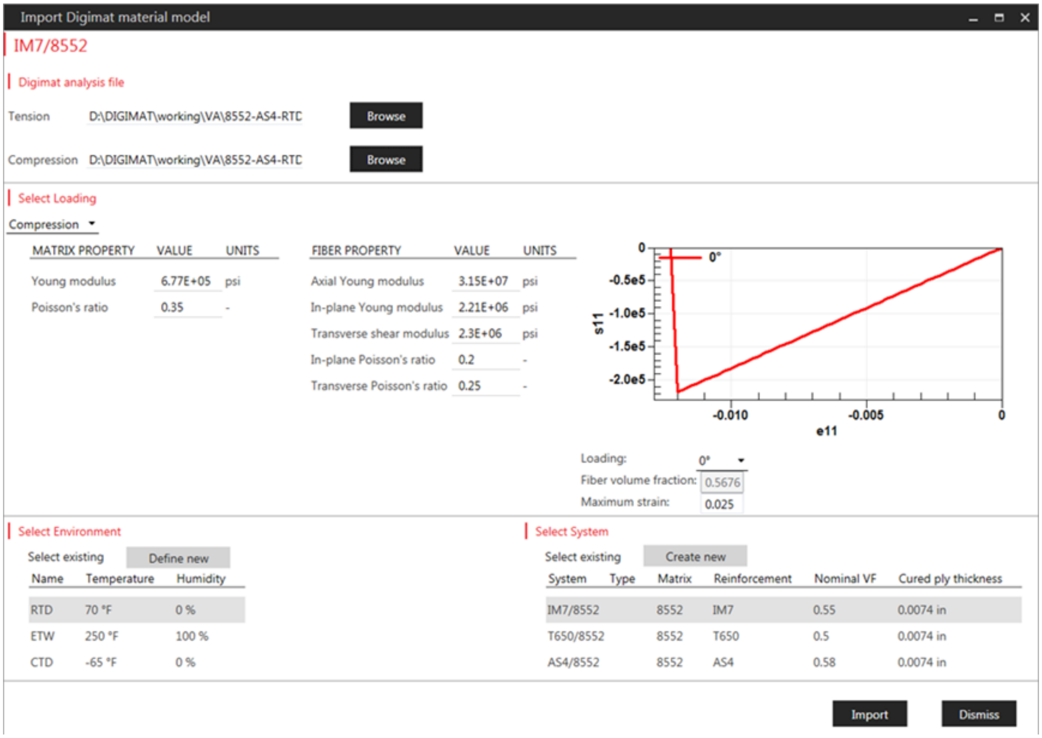


Figure 5-2 Import Digimat model window.

Two Digimat analysis files should be provided: one for the tension behavior of the material and another one for the compression behavior.

The following constraints must be satisfied:

- both files must use the same unit system
- both files must contain a purely mechanical analysis
- both files must contain one matrix phase and at least one continuous fiber phase
- both files must have at least one progressive failure indicator

In the lower part of the window (see Figure 5-2), the environment and system which the imported model should be associated to must be selected. For both environment and system, it is possible to associate to an existing item or to create a new one.

Once the Digimat model has been imported, it can be used in the same way as regular material model calibrated from experimental data. However, the imported models are only valid for the fiber volume fraction indicated in the analysis file and they cannot be used directly in analyses involving variability.

Whenever an attempt is made to use an imported material model in an analysis involving variability, Digimat- VA will try to calibrate a new material model based on the stiffnesses and strength of the imported model. This process is performed automatically and in a transparent way, but it is important to keep in mind that from this point, all subsequent simulations will be performed using the calibrated material model.

Import Experimental Data

In the same way as for Digimat material model, it is possible to import experimental data and associate it to a system and an environment, either new or existing, (see Figure 5-3). After the experimental data is imported, it can be used to calibrate a material model.

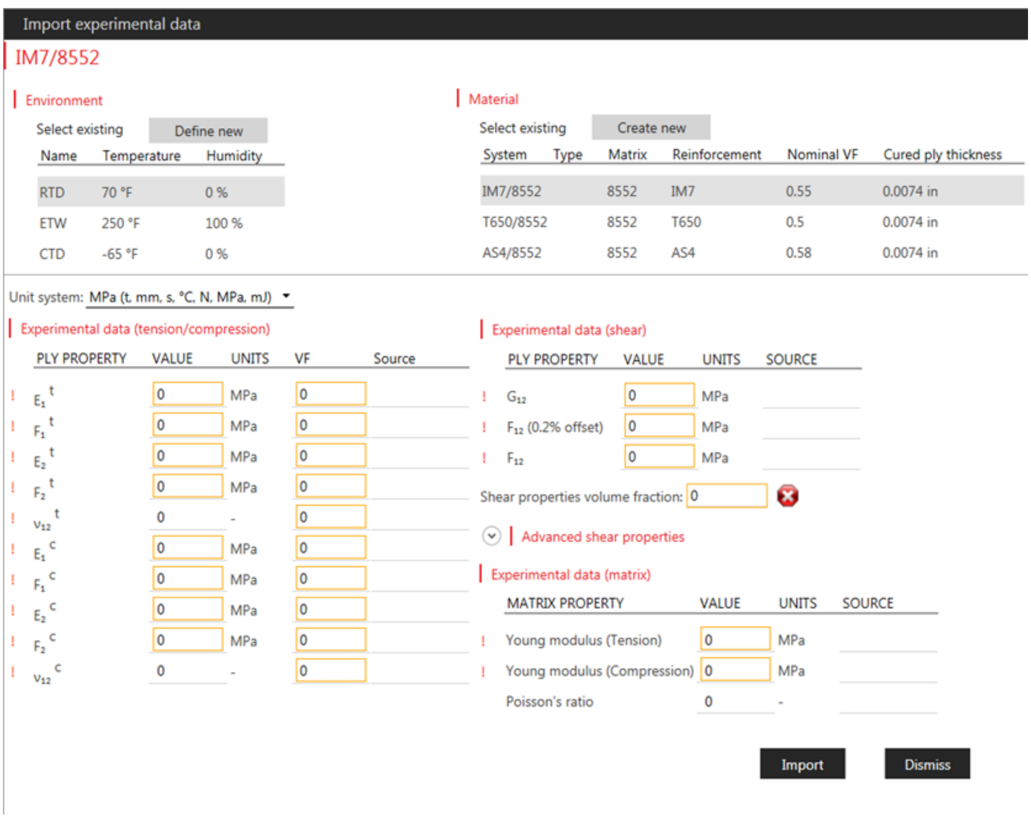


Figure 5-3 Import experimental data window.

Update Digimat-VA Database from Previous Versions

By default when installing Digimat-VA a new database is created upon installation, providing access to all latest improvements compared to previous releases (ex: new type of tests available).

To benefit from data available from an existing Digimat-VA database (ex: existing material data and models for various conditions, or userdefined layups) in a new Digimat-VA installation, there is possibility to upgrade the existing database during the Digimat installation, by browsing to an existing `vadb.db` file from a previous Digimat-VA version.

Interface to MaterialCenter

Digimat-VA proposes an interface to MaterialCenter which serves two purposes:

- In the Material model step, you can import ply properties stored in MaterialCenter for material model calibration. The calibrated per-phase properties can then be stored back in MaterialCenter.
- In the Global post-processing step, you can export laminate virtual allowables and store them in MaterialCenter for data management at the enterprise level.

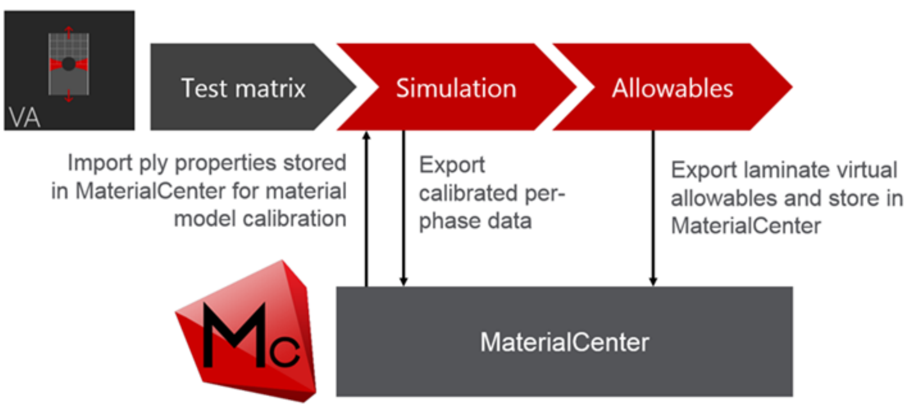


Figure 5-4 Interactions between Digimat-VA and MaterialCenter.

Managing Material Data at Material Model Calibration Step

In order to obtain the experimental data required for the calibration step in Digimat-VA, the following steps must be followed in MaterialCenter:

1. Go to **Navigate** panel.
2. In the tree, select **All Digimat VA** materials.
3. Choose the material in the tree.
4. Double click on the **material line**.

5. In the material section visualization click **Export to Excel.**
6. Choose the window for mapping.
7. In the Schema mapping choose **Imp-Exp Digimat-VA Composite Material** and submit.
8. Download the Excel document. It can then be imported in Digimat-VA in the calibration step.

Once a Digimat model has been calibrated in Digimat-VA, it's possible to export it to MaterialCenter. To import the data in MaterialCenter, the following steps must be followed:

1. Go back in **MaterialCenter** in the material section.
2. Click **Edit.**
3. Choose **Material Excel**.
4. In Excel file to import, select the **Excel file** (e.g., calibrated- material modelVA).
5. Schema mapping: Imp-Exp Digimat-VA CompositeMaterial.

Once the process is completed the **Resin and Fiber** properties appear. The experimental and virtual properties can be compared.

Managing Virtual Allowables Data

When virtual allowables have been computed, they can be stored in MaterialCenter in the following way:

1. In the global post-processing section of Digimat-VA, click **Export to MaterialCenter** to generate the Excel results file
2. Back in the material section in MaterialCenter, go to **Tools.**
3. Click **Import Test Data** and choose the Excel results file (e.g. VA-Basis Results).
4. In Test Template, choose **Import Digimat-VA Composite Tests**, submit and wait for the process to complete.

To visualize the imported data, in the material section go into Test data section and double click on one dataset. The data and the curve will appear.

To import the virtual global results in MaterialCenter, do the following:

1. Use the same Excel file and repeat the same step in the material section
2. Edit / Material Excel
3. In Excel file to import, choose the file containing the global results (e.g., VA-BasisResults.xlsx)
4. In Schema mapping, choose Import Digimat–VA Configurations and submit

In the material section in MaterialCenter, the laminate mechanical properties section will be filled for each layup. Click on the lens to visualize the CoV used in the allowable analysis.

6 Command Line

- Introduction
- Input File Structure

Introduction

The command line mode of Digimat-VA allows to run complete analysis without any graphical interaction. The command line mode uses a different executable (`DigimatVA-Batch.bat`, located in the Digimat-VA installation directory, `DigimatVA\exec`), which takes as input an ASCII file (extension .vaf) containing the whole test matrix definition (described in Input File Structure).

To make the creation of such input file easier, it is possible to export and import `.vaf` file from the user interface (**Save project** or **Open project** in the main menu, then selecting Digimat-VA analysis file (`*.vaf`) as file type). The `*.vaf` file obtained in that way can then be edited manually before being submitted to the command line executable.

The syntax for the command line executable is simply DigimatVA-Batch.bat analysis.vaf. Executing this command will start the command line version of Digimat-VA, load the test matrix defined in the provided `*.vaf` file, then generate all the jobs and then submit and post-process them. A basic progress information is provided on the command prompt. Once all jobs have completed, results are exported and the executable terminates. A log file with the same name as the input `*.vaf` file is generated next to the input file.

Input File Structure

The `*.vaf` file is organized in sections, similarly to a Digimat-MF or Digimat-FE `.daf` file. Each section contains a collection of key-value pairs, using the format key = value

The allowed sections are

- `PROJECT`
- `MATERIAL`
- `LAYUP`
- `CARPET_LAYUP`
- `TEST`
- `ENVIRONMENT`
- `VARIABILITY`
- `MATERIAL_MODEL`
- `FEA_SETTINGS`
- `VIRTUAL_TEST`
- `JOB_SUBMISSION`
- `PARAMETRIC_STUDY`
- `DEFECT`
- `REMOTE_HOST`

In the following, every key which accepts a finite set of values will present all possible values in square brackets, separated by /. Optional key - value pairs will be presented in square brackets. Most sections have a name that allows to uniquely identify them in other related sections.

Project Section

There can only be one project section in a `.vaf` file. It is used to define the unit system of the project and its working directory.

```
##########################################
PROJECT
unit_system = [MPa / MPa-ms / Pa / GPa / CGS / SI / FPS / psi / ksi]
project_directory = C:\MSC.Software\Digimat\working\MyVAProject
```

Material Section

The material section defines a material system to be used in the test matrix (see Material Definition for details). There can be as many material section as needed in a .vaf file.

```
##########################################
MATERIAL
name = AS4/8552-UD
unit_system = [MPa / MPa-ms / Pa / GPa / CGS / SI / FPS / psi / ksi]
type = [UD / balanced_woven / unbalanced_woven]
fiber = AS4
matrix = 8552
nominal_volume_fraction = 0.59
cured_ply_thickness = 0.18796
[warp_weight_rate = 0.5]
```

Layup and Carpet Layup Section

The layup section defines a layup to be used in the test matrix. The stacking key use the same notation as the user interface for stacking definition (see Layup Definition). There can be as many layup section as needed in a `.vaf` file.

```
##########################################
LAYUP
name = Quasi-iso-Example
stacking = [45/0/-45/90]_2s
```

The carpet layup section has the same purpose, but is dedicated to layup for carpet plot generation. The different keys have the same meaning as in the user interface (see Layup Definition).

```
##########################################
CARPET_LAYUP
name = CarpetPlot
first_angle = 0
intermediate_angle = 45
last_angle = 90
layup_proportion_increment = 0.25
```

Test Section

The test section define a standard test (see Standard Test Definition for details). The set of parameters to use is obviously dependent of the type of test. There can be as many test section as needed in a `.vaf` file.

```
##########################################
TEST
name = UNT_ASTM D3039
unit_system = [MPa / MPa-ms / Pa / GPa / CGS / SI / FPS / psi / ksi]
norm = ASTM D3039
type = [UNT / UNC / OHT / OHC / FHT / FHC / IPS45 / IPSVN / SSB2PSFC /
SSB2PSFT / SSB2PSFTS / SSB2PDFC / SSB2PDFT / SSB2PDFTS / SSB1P /
DSBSFT / DSBDFT / B3P / B4P / DWI / TAI / CAI / SBS / CBS / DCB /
CELS / ENF]
length = 254
width = 25.4
```

Environment Section

The environment section defines a set of environmental conditions for the test matrix (see Environment Conditions for details).

```
##########################################
ENVIRONMENT
name = RTD
unit_system = MPa
temperature = 21.1111111111111
humidity = 0
```

Variability Section

The variability section allows to define the variability for the current analysis (see Variability definition for details). The section presented below shows the parameters for standard variability. For the other types of variability, there is no extra parameter.

```
##########################################
VARIABILITY
type = [no_variability / standard / parametric_study / defect_study]
number_of_batches = 3
number_of_panels = 6
number_of_specimens = 3
```

Material Model Section

This section store all the necessary information to build a calibrated Digimat material. It is associated to a material and an environment section (through the material and environment keys). The keys related to experimental ply properties have a name ending in t or c depending if they are related to a tensile or compressive property.

The value associated to those keys is a semicolon separated pair of values where the first value is the experimental property and the second value is the fiber volume fraction at which it was measured. The keys related to matrix or fiber properties have a name starting with matrix or fiber respectively.

The keys related to CTE, CME and densities are only necessary if the CLT analysis flag is toggled on.

```
##########################################
MATERIAL_MODEL
name = AS4/8552-UD_RTD_AS4/8552-UD_RTD_Digimat 3
unit_system = [MPa / MPa-ms / Pa / GPa / CGS / SI / FPS / psi / ksi]
material = AS4/8552-UD
environment = RTD
E1t = 131551.770618052;0.5956
F1t = 2063.04616374914;0.5956
E2t = 9238.9608295697;0.5872
F2t = 63.9143036493367;0.5872
nu12t = 0.302;0.5956
E1c = 115555.957838499;0.6176
F1c = 1484.37005746124;0.6176
E2c = 9859.48804946617;0.6148
F2c = 267.860916588644;0.6148
nu12c = 0.335;0.6176
G12 = 4826.32282141701;0.5885
F12_02 = 55.1579751019087;0.5885
F12 = 91.5622386691684;0.5885
eps12_F12 = 0.05;0.5885
[eps12_break = 0.06;0.5885]
[F12_break = 96.5393840787547;0.5885]
[CTE1 = 0;0]
[CTE2 = 0;0]
[CME1 = 0;0]
[CME2 = 0;0]
[matrix_density = 0]
matrix_young_t = 4667.74364299902
matrix_young_c = 4667.74364299902
matrix_Poisson_ratio = 0.35
[fiber_density = 0]
[fiber_cte_axial = 0]
failure_modelling = [PFA / FPF]
damage_evolution = instantaneous
interface_model = [no_damage / cohesive]
[interface_GI = 0.4]
[interface_GII = 0.6]
[interface_TI = 25]
[interface_TII = 40]
[benzeggah_exponent = 1.6]
[random_seed = 499397452]
```

FEA_Settings Section

The `FEA_SETTINGS` section are optional. If no such section is present in the `.vaf` file, the default settings from the VADB will be used when generating the FEA models.

```
##########################################
FEA_SETTINGS
name = Default settings UNT
test_type = [UNT / UNC / OHT / OHC / FHT / FHC / IPS45 / IPSVN /
```

```
SSB2PSFC / SSB2PSFT / SSB2PSFTS / SSB2PDFC / SSB2PDFT / SSB2PDFTS /
SSB1P / DSBSFT / DSBDFT / B3P / B4P / DWI / TAI / CAI / SBS / CBS /
DCB / CELS / ENF]
reduced_integration = on
symmetry_bc = on
element_in_length = 20
element_in_width = 10
number_time_step = 200
minimum_time_step = 0.00125
applied_strain = 0.018
smart_timestepping = on
stop_at_load_drop = on
only_delaminate_in_main_interface = on
unnotched_modelling_type = full_coupon
```

Virtual Test Section

`VIRTUAL_TEST` sections are optional. Either one `VIRTUAL_TEST` section is defined for each virtual test in the test matrix (i.e. only the tests explicitly defined will be created and ran) or no `VIRTUAL_TEST` section at all. In that last case, all possible combinations are created and ran, with default settings and material model assignments (i.e. similar to what happens in the UI if no test has been manually disabled)

Each `VIRTUAL_TEST` section contains a reference to a material, a layup, a test, an environment, the material model (which has obviously to be defined for the same material and environment) and the FEA settings.

If the type of the `VARIABILITY` section is set to `defect_study`, all defects assigned to the current virtual test must be listed (using the defect key).

```
##########################################
VIRTUAL_TEST
material = AS4/8552-UD
layup = Quasi-iso-Example
test = UNT_ASTM D3039
environment = RTD
material_model = AS4/8552-UD_RTD_AS4/8552-UD_RTD_Digimat 3
fea_setting = Default settings UNT
enabled = on
[defect = defect1]
[defect = defect2]
```

Job Submission Section

The job submission section of the `.vaf` file is unique. It allows to control

- the requested finite element outputs
- the submission type
    - local: it must then contain the `working_directory` and `number_of_solvers` keys.
    - remote: it must then contain a reference to a remote host section

- the outputs to export at the end of the analysis (.xlsx, .docx)

```
###########################################
JOB_SUBMISSION
strain_output = off
stress_output = off
damage_output = on
submission_type = [local / remote / none]
[working_directory = C:\MSC.Software\Digimat\working\ProjectVA_20190701]
[number_of_solvers = 4]
[remote_host = username@192.168.103.120 ]
[project_file = C:\MSC.Software\Digimat\working\project.vcp]
[project_export =
C:\MSC.Software\Digimat\working\ProjectVA_20181116\project.xlsx]
[project_report =
C:\MSC.Software\Digimat\working\ProjectVA_20181116\project.docx]
```

Remote Host Section

For remote run, it is necessary to define the remote host information in this section. The authentication can be either by password or by private key file. Private key file authentication is used if the keyfile key is present in the section. If not, Password authentication is used. For password authentication, the definition of the password in the `.vaf` file is optional. If the password is not present in the vaf file, user will be prompted for password input at the command line when submitting the analysis.

When LSF or PBS are being used, the submission script must be defined at the end of the `REMOTE_HOST` section, each line of the script starting with an **@** character

```
###########################################
REMOTE_HOST
name = user@192.168.103.122
user_name = user
host_name = 192.168.103.122
[port = 22]
[password = password]
[keyfile = C:\Users\me\Documents\SSHKey\key.ppk]
remote_working_directory = /home/me/remoteVAWorkingCmdLine
remote_digimat_install_directory = /opt/Digimat/2023.3
remote_license_env_var = 27500@10.100.103.211
automatic_output_download = on
job_submission_type = [LSF / PBS / direct]
If using direct:
nb_concurrent_run = 1
If using LSF or PBS:
lsf_queue_name = normal
shared_disk_run = on
job_submission_command_line = bsub < %RUN_SCRIPT%
#job submission script
@#
***************************************************************************
*****
@#
@# LSF Queue script to launch Digimat-VA coupon (usage: bsub < run.lsf)
@#
```

```
@#
**************************************************************************
*****
@
@#BSUB -J %JOB_NAME%
@#BSUB -q %QUEUE_NAME%
@#BSUB -W 02:00
@#BSUB -o %JOB_NAME%.o%J
@#BSUB -e %JOB_NAME%.e%J
@
@# *************************** END OF ADAPTABLE SECTION
*************************
@# ************ (The following lines should normally not be modified)
***********
@
@export DIGIMAT_BIN_20233=% REMOTE_DIGIMAT_INSTALL_DIR%/Digimat/exec
@export
LD_LIBRARY_PATH=%REMOTE_DIGIMAT_INSTALL_DIR%/Digimat/lib:$LD_LIBRARY_PATH
@export DIGI2MARC_USUB_LIB_DIR=%PROJECT_DIR%
@export MSC_LICENSE_FILE=27500@192.168.103.211
@
@# Execution command
@%REMOTE_DIGIMAT_INSTALL_DIR%/Digimat/external64/FESolver/tools/run_marc
-j %INPUT_FILE% -v n -q f -ci n -cr n -prog
%REMOTE_DIGIMAT_INSTALL_DIR%/Digimat/external64/FESolver/bin/linux64i8/ma
rcVA.marc
@
@#Force an exit status of 0. Otherwise LSF job will be marked as non
successful.
@exit 0
```

Parametric Study Section

The `PARAMETRIC_STUDY` section is only necessary when the type of the `VARIABILITY` section is set to `parametric_study`. There cannot be more than one such section in a `.vaf` file. Each varying parameter is corresponding to one key - value pair, with the value using either the explicit list notation (semicolon separated) or the range-based notation (three values separated by:). See FE Analysis for details.

```
PARAMETRIC_STUDY
name = param_study_1
type = [full_cross_matrix / single_param_matrix]
matrix_Young = [3000;3500 / 3000:3500:100]
matrix_tensile_strength = ...
fiber_Young = ...
weft_Young = ...
```

Defect Section

The DEFECT sections are only necessary when the type of the `VARIABILITY` section is set to `defect_study`.

```
###########################################
DEFECT
name = waviness1
```

```
type = [waviness / intraply_porosity / interply_porosity /
initial_delamination]
The other key - value pairs are dependent on the defect type.
```

Waviness

```
waviness_type = [uniform / non_uniform / hump / indentation / embedded]
wavelength = 30
position = 120
orientation_angle = 0
mesh_refinement_factor = 8
amplitudes = 0.03;0.04;0.05;0.06;0.07;0.08;0.08;0.08;0.07;0.06;0.05;
0.04;0.03;0.02;0.01;0.0;0.0
```

The value associated to the amplitudes key must be a semicolon separated list of amplitudes, with the number of values equal to the number of plies in the layup of the virtual test this waviness defect is associated with.

Intraply porosity

```
global_defect_flag = [on / off]
[position_x = 100]
[position_y = 20]
[length = 20]
[width = 20]
porosity = 0.03
number_of_porosity = 1
```

Interply porosity

```
global_defect_flag = [on / off]
[position_x = 100]
[position_y = 20]
[length = 20]
[width = 20]
KDF_estimation_type = [user / calibration]
[porosity = 0.03]
[interface_GI_KDF = 0.9]
[interface_GII_KDF = 0.9]
[interface_TI_KDF = 0.9]
[interface_TII_KDF = 0.9]
```

If the `KDF_estimation_type` is set to user, it is necessary to provide the 4 KDF. Otherwise, it is necessary to provide the porosity value for which calibration is to be performed.

Initial delamination

```
global_defect_flag = [on / off]
[position_x = 100]
[position_y = 20]
[length = 20]
[width = 20]
average_delaminated_surface_ratio = 0.15
maximum_delaminated_area = 10
delaminated_interfaces = 1;2;3;4;5;6;7;8;9;10;11;12;13;14;15
```

The `delaminated_interfaces` key is used to specify the list of interfaces where initial delamination must be inserted. It must be a semicolon separated list of integer value between 1 and the number of plies in the layup minus one.

7 Specific Features

- Nonlocal Per-phase Damage Model
- Correction for Out-of-plane Stresses

Nonlocal Per-phase Damage Model

The specific feature `va_enable_micro_pfa` activates the possibility to run coupon simulations in Digimat-VA with material models similar to the models suggested by Wu and coworkers, (2015), (2021), and more recently by Calleja Vázquez and coworkers (2022). In these models, the progressive failure of the fiber material and of the matrix material are modeled separately and combined together through an incrementally-secant mean-field homogenization technique to derive the behavior of the composite material. To avoid significant mesh dependency effects, these progressive failure models can further be enhanced with a nonlocal regularization procedure.

To use such a model in Digimat-VA, the user first needs to create a `.daf` file similar to the one described below and to import it in the material model definition window (see Digimat Material Model Calibration) by clicking on **Create new material model** and selecting **Digimat material (imported)**. Once imported, this material model can be assigned to different coupons and/or supplemented with delamination properties like any other material model in Digimat-VA.

Below are the contents a typical `.daf` file defining a material model with distinct progressive failure mechanisms for the fiber and the matrix materials. To avoid potential issues, it is recommended to only modify the numerical values or character strings located to right of an equal sign in this template file.

```
# USE_NEW_KERNEL

##########################################

MATERIAL

name = MatrixMaterial

type = J2_plasticity

consistent_tangent = on

elastic_model = isotropic

density = 1.301E-09

Young = 4668.0

Poisson = 0.39

yield_stress = 32.0

hardening_model = exponential_law

hardening_modulus = 300.0

hardening_exponent = 100.0

weak_damage_coupling = on

progressive_failure = PFM_Matrix_OFRD


##########################################

MATERIAL
```

```
name = FiberMaterial
type = elastic
elastic_model = transversely_isotropic
density = 1.79E-09
axial_Young = 231000.0
inPlane_Young = 12900.0
inPlane_Poisson = 0.07
transverse_Poisson = 0.3
transverse_shear = 17000.0
progressive_failure = PFM_Fiber_OFRD

##########################################
PHASE
name = MatrixPhase
material = MatrixMaterial
type = matrix
volume_fraction = 0.4

##########################################
PHASE
name = FiberPhase
material = FiberMaterial
type = continuous_fibers
volume_fraction = 0.6
aspect_ratio = 10000
behavior = deformable_solid
orientation = fixed
theta_angle = 90
phi_angle = 0

##########################################
MICROSTRUCTURE
name = Microstructure1
phase = MatrixPhase
```

```
phase = FiberPhase

##########################################
RVE
type = classical
microstructure = Microstructure1

##########################################
LOADING
name = Mechanical
type = strain
load = uniaxial_1
initial_strain = 0
peak_strain = 0.05
history = monotonic
quasi_static = on
strain_rate = 1e-3
theta_load = 90
phi_load = 0

##########################################
ANALYSIS
name = Analysis1
load = DIGIMAT
loading_name = Mechanical
type = mechanical
final_time = 1
number_increment = 200
max_time_inc = 0.01
min_time_inc = 0.001
output_name = output1
homogenization = on
homogenization_model = Mori_Tanaka
second_order = off
```

```
number_angle_increments = 12
homogenization_relative_tol = 0.0001
homogenization_tol = 1E-06
acceptable_homogenization_tol = 0.001
homogenization_it_max = 20
homogenization_monitoring_it = 20
number_angle_increments = 12
hybrid_methodology = off
hybrid_failure_criteria = off
integration_parameter = 0.5
PF_direct_initiation_thoroughness_factor = -1
PF_max_damage = 0.9995
PF_explicit_formulation = on
failure_stop_analysis = off
failure_element_deletion = off

##########################################
FAILURE INDICATOR
name = FailureIndicatorFiber
type = maximum_stress
use_linear_formulation = on
axes = local
tensile_strength = 3630.0
compressive_strength = 2340.0
component = 11

##########################################
PROGRESSIVE FAILURE
name = PFM_Matrix_OFRD
type = lemaitre_damage
damage_initiation_threshold = 0.000000000000000e+00
damage_rate_factor = 725.62358276644
damage_exponent = 2.0
nullify_negative_values = on
```

```
PF_axial_diffusivity = 2.0
PF_inplane_diffusivity = 0.0121

##########################################
PROGRESSIVE FAILURE
name = PFM_Fiber_OFRD
type = single_component_damage
failure_indicator = FailureIndicatorFiber
damage_law_fA = linear_softening,1.0,47.2,0.999,0.999
damage_law_fB = linear_softening,1.0,134.0,0.999,0.999
tensile_to_compression_damage = off
nullify_negative_values = on
PF_axial_diffusivity = 0.01243225
PF_inplane_diffusivity = 0.0

##########################################
OUTPUT
name = output1
RVE_data = Default
Phase_data = All,Default
Engineering_data = None
Log_data = Default
Dependent_data = Default
Fatigue_data = Default
```

The first `MATERIAL` section defines the behavior of the matrix material in the absence of damage. The different types of matrix behavior that can be used and the corresponding syntax are detailed in the section Behavior of the Matrix Material in the Absence of Damage below.

The second `MATERIAL` section defines the behavior of the fiber material in the absence of damage. That behavior is assumed to be transversely isotropic linear elastic. Denoting by 1 the axial direction of the fiber, the values denoted by `axial` in the `MATERIAL` section correspond to component 11, the values denoted by `inPlane` correspond to components 22 or 23 while the values denoted by `transverse` correspond to components 12. See section Linear (thermo-)elasticity for more information about that material behavior.

The first PHASE section primarily serves the purpose of defining the matrix volume fraction which is equal to 40% in the above example while the fiber volume fraction is defined in the second PHASE section and must be equal to 100% minus the matrix volume fraction.

In the ANALYSIS section, the possibility is given to the user to define a cap value for all damage variables in the material model, whether related to the fiber of matrix material, through the PF_max_damage keyword.

In the FAILURE INDICATOR section, the stress-based failure criterion for the fiber material is defined by specifying the values of the axial tensile and axial compression strength.

The damage behavior of the matrix material is defined in the first PROGRESSIVE FAILURE section. The different types of damage behavior that can be used for the matrix material and the corresponding syntax are detailed in the section Damage Behavior of the Matrix Material below. The keywords nullify_negative_values, PF_axial_diffusivity and PF_inplane_diffusivity relate to the nonlocal regularization process and are described in the section Nonlocal Regularization Procedure below.

The damage behavior of the fiber material is defined in the second PROGRESSIVE FAILURE section. Distinct damage evolution laws both producing a stress-strain behavior with linear softening can be defined for the fiber behavior in tensile and in compression. These are defined next to the damage_law_fA and damage_law_fB keywords by means of the linear_softening keyword followed by four comma separated values: f_{min}, f_{max}, D_{max} and D_{final} (see section Linear Softening Damage Evolution Law for the meaning of these values). The keywords nullify_negative_values, PF_axial_diffusivity and PF_inplane_diffusivity are optional and serve the purpose of applying a nonlocal regularization procedure to the failure indicator values for the fiber prior to the computation of damage (see section Nonlocal Regularization Procedure below). The value of these parameters can be different from those defined for the nonlocal regularization of the matrix damage model. The keyword tensile_to_compression_damage, set to off in the above example, means that any damage growth under tensile loads will not affect the compression behavior. A similar keyword named compression_to_tensile_damage can be used to control the coupling in the opposite direction, from damage growth in compression to tensile behavior. Both these parameters are assumed to be set to on by default.

Behavior of the Matrix Material in the Absence of Damage

Two types of matrix behavior are currently offered: the classical J_2 plasticity model and the pressure-dependent plasticity model suggested by Nguyen and co-workers (2016) which is an enhanced version of the Drucker-Prager model. Both can be used with all types of hardening laws available in Digimat-MF for the J_2 plasticity model (see section Elasto-plasticity: J_2-plasticity Model). The syntax of the hardening law definition can be obtained by creating a material model with the desired law in the graphical user interface of Digimat-MF, saving the model to a .daf file and viewing the contents of that file in a text editor. The syntax examples below both make use of the exponential hardening law which obeys the following equation:

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

where p is the accumulated plastic strain and R_{∞} and m are material parameters.

J_2 Plasticity Model

The J_2 plasticity model is described in details in section Elasto-plasticity: J_2-plasticity Model. It must defined in the `.daf` file using the syntax below:

```
MATERIAL
name = MatrixMaterial
type = J2_plasticity
consistent_tangent = on
elastic_model = isotropic
density = 1.301E-09
Young = 4668.0
Poisson = 0.39
yield_stress = 32.0
hardening_model = exponential_law
hardening_modulus = 300.0
hardening_exponent = 100.0
weak_damage_coupling = on
progressive_failure = PFM_Matrix_OFRD
```

where:

- `density` is the density of the material.
- `Young` is the Young's modulus.
- `Poisson` is the Poisson ratio.
- `yield_stress` is the initial yield stress σ_Y
- `hardening_modulus` is the parameter R_{∞} in the hardening law
- `hardening_exponent` is the parameter m in the hardening law

Nguyen's Pressure-dependent Plasticity Model

In the pressure dependent plasticity model suggested by Nguyen and co-workers (2016), the yield surface is defined as follows:

$$\Phi(\sigma_{eq}, \sigma_m, p) = \left(\frac{\sigma_{eq}}{\sigma_c}\right)^{\alpha} + 3\frac{M^{\alpha} - 1}{M + 1}\frac{\sigma_m}{\sigma_c} - \frac{M^{\alpha} + M}{M + 1}$$

whereas the flow potential is defined as follows:

$$G(\sigma_{eq}, \sigma_m) = \sigma_{eq}^2 + \frac{9}{2}\frac{1 - 2v_p}{1 + v_p}\sigma_m^2$$

where:

- σ_{eq} is the equivalent von Mises stress
- $\sigma_m = -(\sigma_{11} + \sigma_{22} + \sigma_{33})/3$ is the hydrostatic pressure
- $\sigma_c = \sigma_y + R(p)$ is the current yield stress in compression
- M is a material parameter defining the ratio between the current yield stress in tension and the current yield stress in compression
- α is a material parameter
- v_p is a material parameter corresponding to the plastic Poisson ratio at the onset of plastic deformation

This pressure-dependent plasticity model can be defined in the `.daf` file using the syntax below:

```
##########################################
MATERIAL
name = MatrixMaterial
type = Drucker_Prager_Nguyen
consistent_tangent = on
elastic_model = isotropic
density = 1.301E-09
Young = 4668.0
Poisson = 0.39
yield_stress = 32.0
hardening_model = exponential_law
hardening_modulus = 300.0
hardening_exponent = 100.0
yield_function_exponent = 2
```

```
yield_stress_ratio = 0.49013502
plastic_Poisson_ratio = 0.33
isotropic_method = spectral
weak_damage_coupling = on
progressive_failure = PFM_Matrix_OFRD
```

where:

- `density` is the density of the material
- `Young` is the Young's modulus
- `Poisson` is the Poisson ratio
- `yield_stress` is the initial yield stress in compression σ_y
- `hardening_modulus` is the parameter R_∞ in the hardening law
- `hardening_exponent` is the parameter m in the hardening law
- `yield_function_exponent` is the parameter α in the yield surface
- `yield_stress_ratio` is the parameter M in the yield surface
- `plastic_Poisson_ratio` is the parameter v_p in the flow potential

Damage Behavior of the Matrix Material

Four types of damage behavior are currently offered for the matrix material. They are all based on the value of the accumulated plastic strain, p. In each of these damage models, damage is predicted to grow if and only if the accumulated plastic strain is larger than a threshold value, p_{th}. That threshold value can be either constant or dependent upon the triaxiality η which is the ratio of the hydrostatic stress to the equivalent von Mises stress:

$$\eta = \frac{(\sigma_{11} + \sigma_{22} + \sigma_{33})/3}{\sigma_{eq}}$$

For all damage models except Nguyen's damage model which relies on specific definition of p_{th}, a constant value of the threshold accumulated plastic strain is defined as follows in the `PROGRESSIVE_FAILURE` section of the `.daf` file:

```
damage_initiation_threshold = 1.000000000000000e-02
```

while a triaxiality dependent value is defined by means of three values corresponding to uniaxial compression ($\eta = -1/3$), pure shear ($\eta = 0$) and uniaxial tension ($\eta = 1/3$) stress states:

```
damage_initiation_threshold_compression = 0.1
```

```
damage_initiation_threshold_shear = 0.03
damage_initiation_threshold_tensile = 0.01
```

The threshold value at any triaxiality value is then obtained from these three values (see Figure 7-1):

- using linear extrapolation between the values at $\eta = -1/3$ and $\eta = 0$ for $\eta < -1/3$
- using piecewise linear interpolation for $-1/3 \le \eta \le 1/3$
- using a constant value equal to the value at $\eta = 1/3$ for $1/3 < \eta$

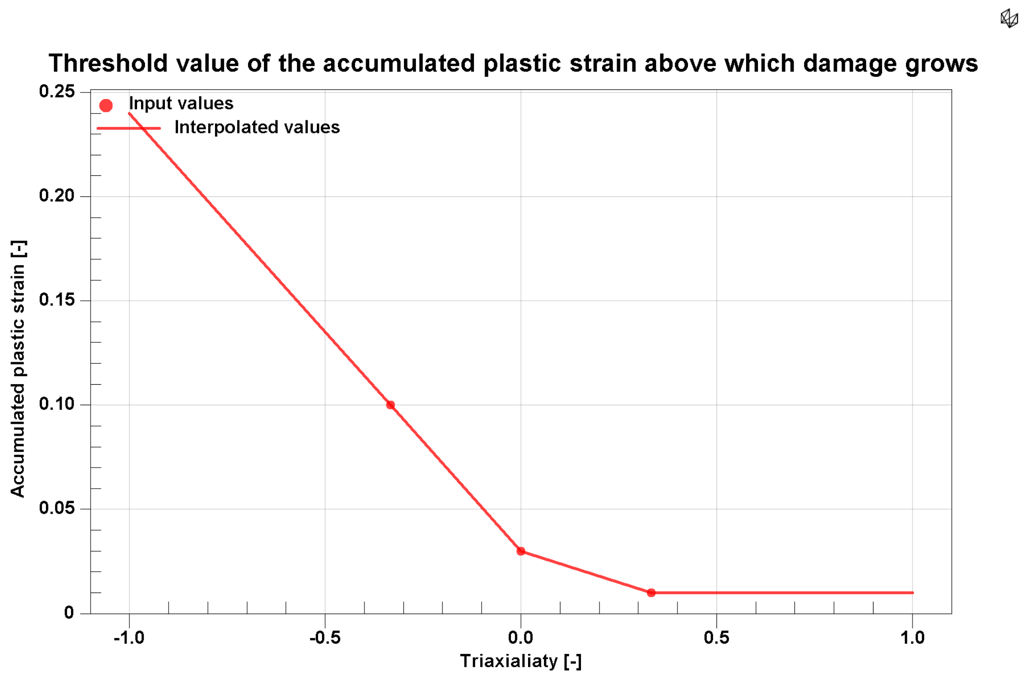


Figure 7-1 Threshold value of the accumulated plastic strain above which damage is predicted to grow as a function of the triaxiality η based on the three input values at $\eta = -1/3$, 0 and $1/3$

Lemaitre Damage Model

The Lemaitre damage model is described in details in section Elasto-plastic Damage Material. In this model, the damage growth rate $\dot{D}$ is computed from the equation below:

$$\dot{D} = \frac{\sigma_Y}{S_0}\left(\frac{Y}{\sigma_Y}\right)^n \dot{\gamma}$$

where:

- σ_y is the initial yield stress
- Y is the strain energy release rate which can be computed from the current stress tensor and the elastic properties of the material
- γ is the largest value of $[p - p_{th}(\eta)]$ encountered since the beginning of the simulation:

  $\gamma = max_{0 \le \tau \le t}[p(\tau) - p_{th}(\eta(\tau))]$
- p is the accumulated plastic strain
- $p_{th}(\eta)$ is the threshold value of the accumulated plastic strain above which damage is predicted to grow which can depend upon the triaxiality η
- S_0 and n are material parameters

The Lemaitre damage model can be defined in the .daf file using the syntax below:

```
##########################################
PROGRESSIVE FAILURE

name = PFM_Matrix_OFRD

type = lemaitre_damage

damage_initiation_threshold_compression = 0.1

damage_initiation_threshold_shear = 0.03

damage_initiation_threshold_tensile = 0.01

damage_rate_factor = 725.62358276644

damage_exponent = 2.0

nullify_negative_values = on

PF_axial_diffusivity = 2.0

PF_inplane_diffusivity = 0.0121
```

where:

- the three `damage_initiation_threshold_xxx` values define the threshold accumulated plastic strain p_{th} as a function of the triaxiality η as explained at the beginning of the section Damage Behavior of the Matrix Material.
- `damage_rate_factor` is the value of $1/S_0$ in the equation for the damage growth rate

- `damage_exponent` is the parameter η in the equation for the damage growth rate
- `nullify_negative_values`, `PF_axial_diffusivity` and `PF_inplane_diffusivity` are optional keywords which define the nonlocal regularization procedure to be applied as explained in a dedicated section below

Lemaitre Damage Model with Saturation

A modified version of the Lemaitre damage model is also offered. In this model, past a certain threshold damage value, the damage does not follow the classical Lemaitre damage model but rather saturates asymptotically at a user-defined value as the accumulated plastic strain continues to grow. This modification was shown to ease the convergence of the finite element simulations.

The modified Lemaitre damage model can be defined in the `.daf` file using the syntax below:

```
###########################################
PROGRESSIVE FAILURE

name = PFM_Matrix_OFRD

type = lemaitre_damage_with_saturation

damage_initiation_threshold_compression = 0.1

damage_initiation_threshold_shear = 0.03

damage_initiation_threshold_tensile = 0.01

damage_rate_factor = 725.62358276644

damage_exponent = 2.0

damage_saturation_threshold = 0.5

damage_at_saturation = 0.99

nullify_negative_values = on

PF_axial_diffusivity = 2.0

PF_inplane_diffusivity = 0.0121
```

where `type` needs to be set to `lemaitre_damage_with_saturation` and two extra material parameters need to be defined compared to the classical Lemaitre damage model:

- `damage_saturation_threshold`, the damage value above which the model differs from the classical Lemaitre damage model and above which damage starts saturating asymptotically at a user-defined value.
- `damage_at_saturation`, the damage value at which the damage will asymptotically saturate.

Exponential Function of the Accumulated Plastic Strain

In this model, the damage variable D is an exponential function of the accumulated plastic strain which will asymptotically reach a user-defined damage value D_{max}:

$$D = D_{max}[1 - exp(-m\gamma)]$$

where:

$$\gamma = max_{0 \le \tau \le t}[p(\tau) - p_{th}(\eta(\tau))]$$

and m is a material parameter controlling the rate at which D will reach D_{max} while γ, p and p_{th} (η) have the same meaning as in the Lemaitre damage model.

This damage model can be defined in the `.daf` file using the syntax below:

```
##########################################
PROGRESSIVE FAILURE
name = PFM_Matrix_OFRD
type = exponential_accpstrain
damage_initiation_threshold_compression = 0.1
damage_initiation_threshold_shear = 0.03
damage_initiation_threshold_tensile = 0.01
damage_rate_factor = 725.62358276644
damage_at_saturation = 0.99
nullify_negative_values = on
PF_axial_diffusivity = 2.0
PF_inplane_diffusivity = 0.0121
```

where:

- the three `damage_initiation_threshold_xxx` values define the threshold accumulated plastic strain p_{th} as a function of the triaxiality η as explained at the beginning of the section Damage Behavior of the Matrix Material.

- `damage_rate_factor` is the parameter m which controls the rate at which the damage will grow.
- `damage_at_saturation` is the damage value D_{max} that will be asymptotically reached by the damage variable.
- `nullify_negative_values`, `PF_axial_diffusivity` and `PF_inplane_diffusivity` are optional keywords which define the nonlocal regularization procedure to be applied as explained in a dedicated section below.

Nguyen Damage Model

Nguyen and co-workers (2019) proposed a damage model function of the accumulated plastic strain. In their model:

- damage does not grow until the accumulated plastic p strain reaches a threshold value p_{th} which is an exponentially decreasing function of the triaxiality η
- below a certain damage value, the damage growth rate is function of both the damage variable and the difference between the accumulated plastic strain and the above threshold value
- past that particular damage value, damage growth slows down and the damage variable asymptotically reaches a value of 1 which facilitates the convergence of the finite element simulations

The model of Nguyen and co-workers (2019) was modified so that the damage would asymptotically reach a user defined value D_{max} potentially smaller than 1 instead of a value of 1. With this modified damage model, damage is computed using the equation below:

$$D = 1-\left[1-\left(\frac{\gamma_{eff}}{\gamma_c}\right)^{\zeta_f+1}\right]^{\frac{1}{\zeta_d+1}}$$

with:

$$\gamma_{eff} = \left\{\begin{matrix} \gamma \\ \gamma_{th} + (\gamma_{max}-\gamma_{th})\left[1-\exp\left(-\frac{\gamma-\gamma_{th}}{\gamma_{max}-\gamma_{th}}\right)\right]\end{matrix}\right\}\begin{matrix} if(\gamma \le \gamma_{th}) \\ otherwise\end{matrix} \quad \text{and}$$

$$\gamma = max_{0 \le \tau \le t}[p(\tau) - p_{th}(\eta(\tau))]$$

where:

- $\gamma_c = \left(\frac{1}{H}\frac{\zeta_f+1}{\zeta_d+1}\right)^{\frac{1}{\zeta_f+1}}$ is the value of γ_{eff} at which the damage will reach a value of 1

- H, ζ_f and ζ_d are material parameters which control the damage growth rate
- $p_{th}(\eta) = a\ exp(-b\eta)+c$ is the threshold value of the accumulated plastic strain p_{th} above which damage is predicted to grow and is a function of the triaxiality η
- a, b and c are material parameters which control the evolution of that threshold accumulated plastic strain with the triaxiality
- $\gamma_{max} = \gamma_c\left[1-(1-D_{max})^{\zeta_d+1}\right]^{\frac{1}{\zeta_f+1}}$ is the value of γ_{eff} at which the damage will reach a value of D_{max}
- D_{max} is a material parameter defining the maximum value of damage that will be asymptotically reached
- $\gamma_{th} = r\gamma_{max}$ is the value of γ above which damage growth starts slowing down until damage will asymptotically reach a value equal to D_{max}
- r is a material parameter controlling the onset of damage growth rate reduction

This modified Nguyen damage model can be defined in the .daf file using the syntax below:

```
##########################################
PROGRESSIVE FAILURE

name = PFM_Matrix_OFRD

type = Nguyen

damage_initiation_factor = 0.022331808

damage_initiation_exponent = 1.543133042

damage_initiation_offset = 0.002648007

damage_rate_factor = 180.0

damage_exponent_f = 0.3

damage_exponent_d = 0.4

damage_saturation_threshold = 0.8

damage_at_saturation = 0.99

nullify_negative_values = on

PF_axial_diffusivity = 2.0

PF_inplane_diffusivity = 0.0121
```

where:

- `damage_initiation_factor`, `damage_initiation_exponent` and `damage_initiation_offset` are respectively the parameters a, b and c which control the evolution of the threshold accumulated plastic strain p_{th} as a function of the triaxiality η
- `damage_rate_factor`, `damage_exponent_f` and `damage_exponent_d` are respectively the parameters H, ζ_f and ζ_d which control the damage growth rate
- `damage_saturation_threshold` is the parameter $r = \gamma_{th} / \gamma_{max}$ which controls the onset of damage growth rate reduction
- `damage_at_saturation` is the damage value D_{max} that will be asymptotically reached by the damage variable
- `nullify_negative_values`, `PF_axial_diffusivity` and `PF_inplane_diffusivity` are optional keywords which define the nonlocal regularization procedure to be applied as explained in a dedicated section below

Nonlocal Regularization Procedure

Due to damage, the material response (either matrix or fiber) can exhibit a softening behavior which makes the solution of the finite element problem significantly mesh dependent. This mesh dependency can largely be reduced by applying a nonlocal regularization procedure as explained e.g. Wu and corworkers (2015). This procedure consists in smoothing the spatial variations of the variable controlling the evolution of damage prior to computing the damage values. Denoting by θ the local value of that variable and by $\tilde{\theta}$ its nonlocal or smoothed value, this smoothing is achieved by solving the following diffusion equation for $\tilde{\theta}$:

$$\tilde{\theta} - c_{11}\frac{\partial^2\tilde{\theta}}{\partial x_1^2} - c_{22}\frac{\partial^2\tilde{\theta}}{\partial x_2^2} = \theta$$

where:

- x_1 denotes the longitudinal direction of the fibers and c_{11} is the corresponding diffusivity value
- x_2 denotes the in-plane direction which normal to the fiber direction (i.e. the transverse direction) and c_{22} is the corresponding diffusivity value

The out-of-plane or thickness direction, x_3, is absent from that equation meaning that the local values are not diffused from one ply to the other by only within the individual plies.

For the damage behavior of the matrix, the variable controlling the damage evolution which needs to be made nonlocal (i.e. smoothed) is:

$$\gamma = max_{0 \le \tau \le t}[p(\tau) - p_{th}(\eta(\tau))]$$

For the damage behavior of the fibers, the variables which need to be made nonlocal are the values of the failure criterion in tension and in compression.

To activate the nonlocal regularization procedure, the following keywords need to be added to the .daf file in the `PROGRESSIVE FAILURE` section defining the damage behavior:

```
nullify_negative_values = on
PF_axial_diffusivity = 2.0
PF_inplane_diffusivity = 0.0121
```

where:

- `PF_axial_diffusivity` is the value of the diffusivity in the longitudinal direction of the fibers c_{11}
- `PF_inplane_diffusivity` is the value of the diffusivity in the transverse direction c_{22}
- `nullify_negative_values` can be set equal to `on` or `off` and, when activated, implies that the local values of the variable controlling the damage evolution which are negative are set to zero prior to applying the nonlocal regularization procedure. This is only making a difference for the matrix damage since the variables controlling the evolution of fiber damage are always positive. It tends to produce higher nonlocal values. It is recommended to set this parameter equal to `on` which is therefore the default behavior.

The nonlocal regularization procedure can be activated independently for the fibers and for the matrix by adding the three above keywords to the `PROGRESSIVE FAILURE` section defining the damage behavior of the fibers and of the matrix. Moreover, these parameters can take different values for the matrix and for the fibers.

Correction for Out-of-plane Stresses

The behavior of the Advanced PFA model is unaffected by the presence of out-of-plane stress components: σ_{13}, σ_{23}, σ_{33}. This simplifying assumption is perfectly valid in coupons exhibiting negligible out-of-plane stresses such as the unnotched or open-hole coupons. It can however yield to inaccurate predictions in other coupons such as the filled-hole coupons in which the pretension load in the bolt gives rises to significant out-of-plane compression stresses in the composite region under the washer. Furtado et al. (2019) therefore suggested some modifications to make to the material model to account for the effect these out-of-plane stresses can have on the damage and failure of the composite material. The specific feature `va_enable_out_of_plane_stresses` offers the possibility to use that correction in any coupon using the Advanced PFA model.

Once the specific feature is activated, the user is required to go through the three following steps to effectively use the correction in the coupon simulations:

1. Check the box **Behavior under out-of-plane stresses** in the **Experimental data** tab of the Material Model Definition window and input the values of the biaxial tensile strength F_2t_3t and the biaxial compression strength F_2c_3c (see Figure 7-2).

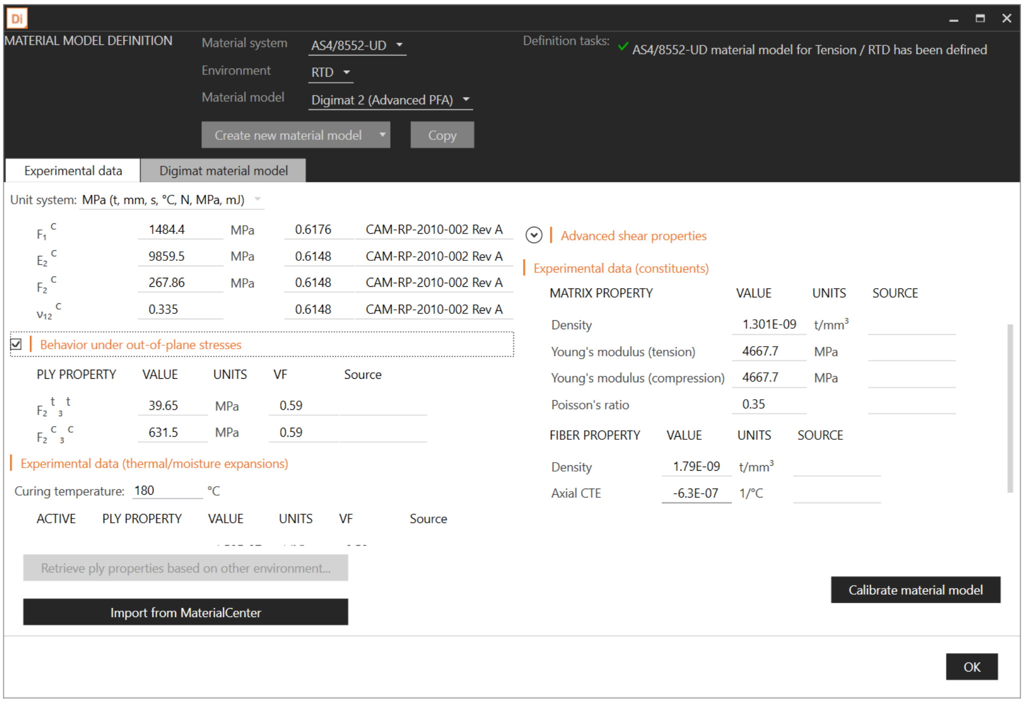


Figure 7-2 Definition of the behavior under out-of-plane stresses in the Experimental data tab.

2. Upon successful calibration of the material model, define the values of ff_XC and eta_G in the **Digimat material model** tab of the **Material Model Definition window** (see Figure 7-3). The frictional parameter in axial compression (ff_XC) controls how the inflection point of the stress-strain curve under uniaxial compression in the fiber direction is affected by the out-of-plane stresses. The enhancement factor for in-plane shear toughness (eta_G) controls how the latter toughness is affected by the out-of-plane stresses.

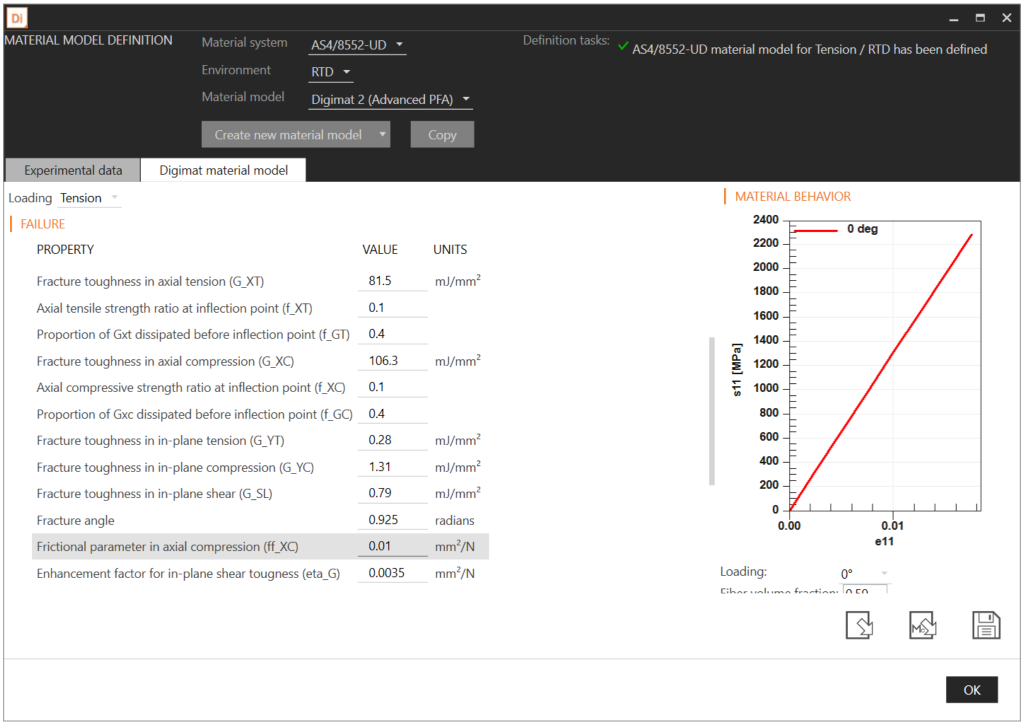


Figure 7-3 Definition of the additional model parameters in the Digimat material model tab.

3. Check the box **Correction for out-of-plane stresses (if applicable)** in the FEA settings (see Figure 7-4) assigned to the coupons for which the correction is requested. This box is checked in the default Advanced PFA settings for filled-hole coupons. For all other coupons, it is not checked in the default Advanced PFA settings so that new settings need to be defined to be able to check the box and activate the correction.

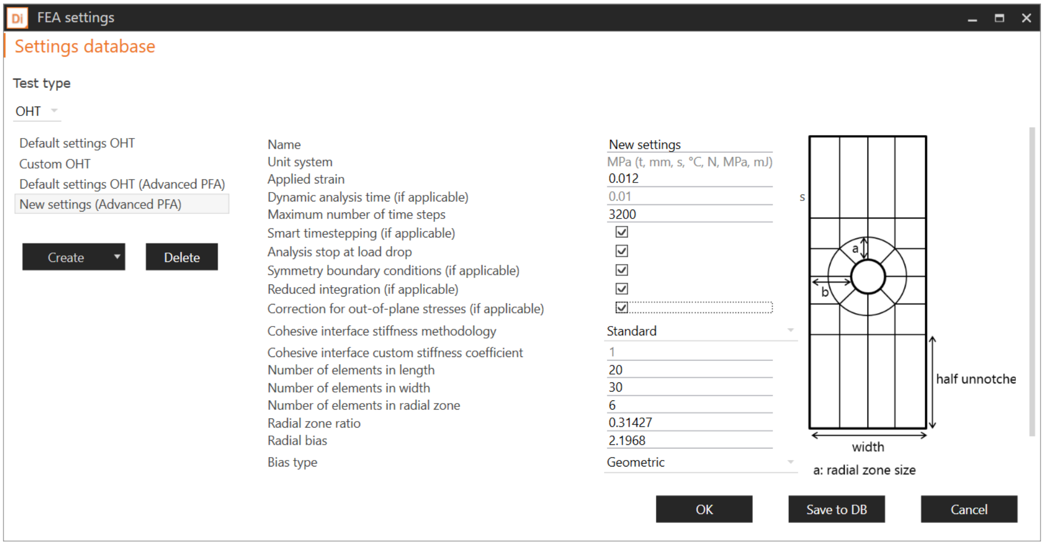


Figure 7-4 Activation of the correction for out-of-plane stresses in the FEA settings.

Note:

1. All three steps above are mandatory. Assigning to a coupon a FEA setting having the box **Correction for out-of-plane stresses (if applicable)** checked (Step 3) will have no effect if the extra material parameters defining the correction (Steps 1 and 2) have not been defined.
2. When using the Advanced PFA model with the correction activated, two extra matrix properties are identified upon successful calibration of the Digimat material model: the biaxial tensile strength and the biaxial compression strength. It is possible to include these two extra matrix properties in a variability study or in a parametric study.

8 Known Limitations

- Material model
- Defect study
- Solver
- Report

Material model

The Advanced PFA model cannot be used with:

- Materials which are not unidirectional
- Bearing tests
- Mono-element tests

Imported Digimat material cards must have:

- UD or woven architecture for the reinforcements
- A progressive failure model

Delamination is not supported for hybrid layups.

Defect study

Defect studies can only be carried out:

- on unidirectional composite materials
- with the Standard PFA material model unnotched, open-hole and filled-hole
- with the Advanced PFA model unnotched and open-hole and no porosity

Solver

- The only supported external solver is Abaqus/Explicit (version 2023).
- Drop weight impact, tension after impact and compression after impact tests are only supported by the external solver.
- Mono element simulations are only supported with the internal solver.

Report

- The preview of the report requires Microsoft Word 2010 (or a newer version) to be installed.

References

1. Composite materials handbook (CMH-17), volume 3. ASTM International, West Conshohocken, Pennsylvania, USA, 2009.
2. J.M. Calleja Vázquez, L. Wu, V.D. Nguyen and L. Noels. An incremental-secant mean-field homogenization model enhanced with a non-associated pressure-dependent plasticity model. International Journal for Numerical Methods in Engineering, 123(19):4616-4654, 2022.
3. C. Chamis, R. Lark, and J. Sinclair. Integrated theory for predicting the hygrothermomechanical response of advanced composite structural components. Advanced composite materials - Environmental effects. ASTM STP 658., pages 160–192, 1978.
4. C. Furtado, G. Catalanotti, A. Arteiro, P.J. Gray, B.L. Wardle and P.P. Camanho. Simulation of failure in laminated polymer composites: Building-block validation. Composite Structures, 226:111168, 2019.
5. V.D. Nguyen, F. Lani, T. Pardoen, X. Morelle and L. Noels. A large strain hyperelastic viscoelastic-viscoplastic-damage constitutive model based on a multi-mechanism non-local damage continuum for amorphous glassy polymers. International Journal of Solids and Structures, 96:192-216, 2016.
6. V.D. Nguyen, L. Wu and L. Noels. A micro-mechanical model of reinforced polymer failure with length scale effects and predictive capabilities. Validation on carbon fiber reinforced high-crosslinked RTM6 epoxy resin. Mechanics of Materials, 133:193-213, 2019.
7. A. Turon, E.V. González, C. Sarrado, G. Guillamet and P. Maimí. Accurate simulation of delamination under mixed-mode loading using a cohesive model with a mode-dependent penalty stiffness. Composite Structures, 184:506-511, 2018.
8. L. Wu, F. Sket, J.M. Molina-Aldareguia, A. Makradi, L. Adam, I. Doghri, I. and L. Noels. A study of composite laminates failure using an anisotropic gradient-enhanced damage mean-field homogenization model. Composite Structures, 126, 246-264, 2015.
9. L. Wu, T. Zhang, E. Maillard, L. Adam, Ph. Martiny and L. Noels. Per-phase spatial correlated damage models of UD fibre reinforced composites using mean-field homogenisation; application to notched laminate failure and yarn failure of plain woven composites. Computers and Structures, 257:106650, 2021.