correlated SOLUTIONS

Application Note

Comparing DIC & FEA

VIC-3D 11

2026

Scope of Support for FEA Integration

This application note describes the integration and data transfer between VIC-3D Digital Image Correlation (DIC) software and third-party Finite Element Analysis (FEA) platforms, specifically Abaqus and Ansys.

While we provide tools to facilitate the export and alignment of DIC data with FEA meshes, please review the following boundaries regarding our technical support.

What We Support

Our technical support team is fully equipped and happy to assist you with:

- ✓ The configuration and operation of VIC-3D.
- ✓ The extraction and formatting of DIC displacement/strain data.
- ✓ Troubleshooting the specific interface tools, extensions, or export scripts provided by Correlated Solutions, Inc.

What We Don't Support

To ensure our team can provide the highest quality assistance on our core product, the design, construction, and troubleshooting of FEA models are strictly beyond the scope of our technical support. Specifically, our team **cannot** provide assistance with:

- Creating geometries, meshing, or partitioning within Abaqus or Ansys.
- Troubleshooting convergence errors, solver warnings, or software crashes within the FEA environment.
- Validating or interpreting the final simulation results.
- *Defining material models, boundary conditions, contact interactions, or load steps (future development will create options for this).*

For assistance with FEA model building and troubleshooting, please refer to the official documentation, user forums, or technical support channels provided by Dassault Systèmes (Abaqus) or Ansys, Inc.

Disclaimer: This application note is provided "as-is" for informational purposes only. Correlated Solutions, Inc. makes no warranties, express or implied, regarding the accuracy or performance of third-party FEA software. Users are solely responsible for verifying the validity, setup, and convergence of their respective finite element models.

Additional Resources

This document does not provide a step-by-step walkthrough of DIC (test set-up, calibration, image acquisition, normal post-processing, etc.). For a range of resources focused on DIC acquisition and analysis, please visit the Correlated Solutions Support Page at www.correlatedsolutions.com/support or click here. ⟶

Comparing DIC & FEA

Introduction

This document outlines the recommended workflow for quantitatively comparing full-field measurements from VIC-3D 11 with the simulated results from a finite element (FE) model. Carefully following these procedures in VIC-3D helps ensure the resulting differences between the DIC and FEA data reflect real measurement-vs-model discrepancies that can be used for validation or model updating (e.g., FEMU).

Before the step-by-step instructions, this document provides a brief overview of two most common causes of failure when comparing DIC and FEA data: Spatial Misalignment and Temporal/Load-state Misalignment.

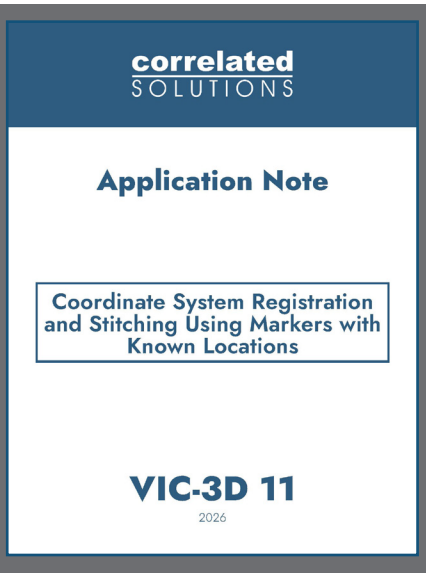
Spatial Alignment

Experimental DIC data and numerical FE results are inherently defined in disparate coordinate frames: the stereo-camera calibration system and the global model coordinate system, respectively. Quantitative comparison requires explicit spatial registration to align these domains. This can be accomplished through two transformation techniques: **Marker-Based Registration** and **Specimen-Geometry Features**.

Marker-Based Registration

The most robust alignment method is Marker-Based Registration. However, it relies on markers on the specimen at locations whose coordinates are known in the FE model frame.

A detailed discussion of marker-based registration can be found in the knowledgebase article: *Coordinate System Registration and Stitching Using Markers with Known Locations*.



Specimen-Geometry Features

Unique features provide another opportunity for alignment of DIC/FE domains. The new **Mesh Alignment Workspace** in VIC-3D (covered later in this document) provides the tools to easily deploy corners, edges, or holes present in the DIC data and FE model to determine the transformation.

Exporting FEA Results

It is important to remember that DIC is a surface measurement. When exporting data from the FE program, it is vital to export the output from the matching surface. This often takes the form of the surface nodes of a solid element, or the integration point of shell elements.

> When registering DIC data to an external coordinate system, **the Best Plane-Fit option in the analysis dialog (under the Post-Processing tab) SHOULD BE TURNED OFF**. Leaving it on is by far the most common single failure mode in this workflow and will silently corrupt the alignment.

Temporal/Load-State Alignment

For quasi-static testing, "temporal" alignment means that the DIC frame and the FE model should have matching mechanical states. At this point, a decision should be made about whether the FE model is driving the DIC test parameters, or whether the FE model will use defined load steps that come from the DIC test.

For the example data provided with this application note, a DIC test was performed on an aluminum dog bone specimen while acquiring images at 4 Hz. The recorded loads from each of the 52 files were then used to create the FE model, creating a load state within the model to match 1:1 with each DIC frame.

When the two sides cannot be matched 1:1, the DIC data can be linearly interpolated between adjacent frames to land on the chosen FE load level. Either way, the load axis — not the clock — is the comparison axis.

Matching Output Definitions

Three details — **Strain Tensor**, **Strain Frame**, and **Smoothing** — account for most mismatches in practice. A field-to-field comparison is only meaningful when both sides report the same quantity.

1. Strain Tensor

VIC-3D offers six strain tensor options (shown below) in the **Calculate Strain** dialog. The FE post-processor must be set to the same definition.

- **Lagrange (default)**
- **Hencky**
- **Euler–Almansi**
- **Logarithmic Euler–Almansi**
- **Engineering**
- **Biot**

PLEASE NOTE: *The "Engineering" strain in VIC-3D is not the raw displacement gradient `dU/dX`; rather, it is derived from the Lagrange tensor (shown to the right) so that it remains insensitive to rigid-body rotation.*

$$\epsilon_x = \sqrt{(1+2E_{xx})} - 1$$

$$\epsilon_y = \sqrt{(1+2E_{yy})} - 1$$

$$\epsilon_{xy} = \sin^{-1}\left(\frac{2E_{xy}}{\sqrt{(1+2E_{xx})(1+2E_{yy})}}\right)$$

2. Strain Frame

VIC-3D computes strain in the local tangential plane of the surface, not in the global X-Y-Z frame. For flat specimens this is a non-issue, but for curved surfaces the FE tensor must be transformed into the local surface frame before any component-wise comparison.

3. Smoothing

VIC-3D computes strain in two passes: it differentiates locally, fitting a deformation gradient to the triangles formed by each data point and its four immediate neighbors, then smooths the resulting exx, eyy and exy over the strain filter window in a separate pass. The filter window is an N × N square of data points — not pixels — with Gaussian weighting, and it acts as a low-pass spatial filter with no direct FE equivalent. Either keep the filter size small enough that the gradients of interest survive, or apply equivalent nodal smoothing to the FE output for a fair comparison.The region that affects a single strain value — the filter window, plus the subsets of the points at its edge — is the virtual strain gauge:

$$\text{VSG [px]} = (L - 1) \times \text{step} + \text{subset}$$

iDICs Good Practices Guide, Ed. 2, Eqn 7.3.

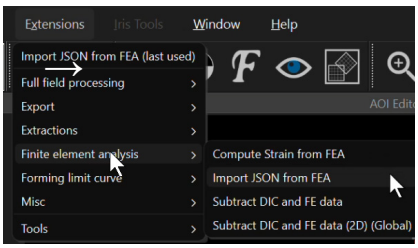
At L = 15, step = 5 px, subset = 21 px and 20 px/mm, the VSG is 4.55 mm — one DIC strain value spanning roughly nine 0.5 mm elements. Expect disagreement at any feature smaller than the VSG.

Strain - recomputed & leveled in Step 5 is the second option above. It applies one filter size and one tensor to the DIC and FE fields in the same run, so no separate nodal smoothing of the FE output is required.

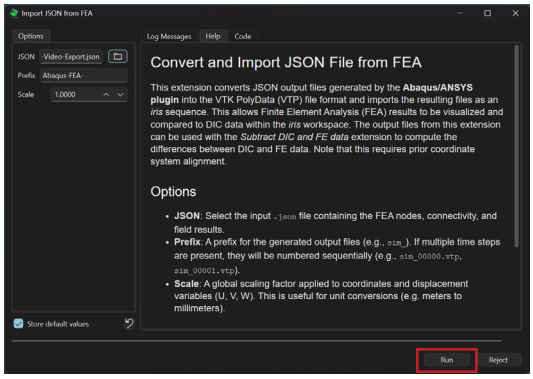
PROCEDURE

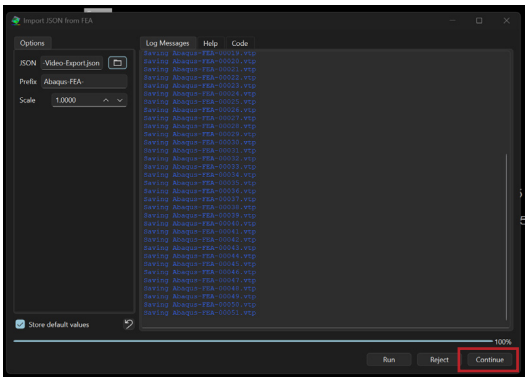
Step 1. Convert FE JSON files into VTP files

After calibration and anaysis (with **Best Plane Fit** turned OFF), the first step is to use the VIC-3D 11 extension workspace to **Convert and Import** the JSON file that was exported from the FEA software. This fully-integrated extension converts the single JSON file into individual VTP files for each DIC frame.

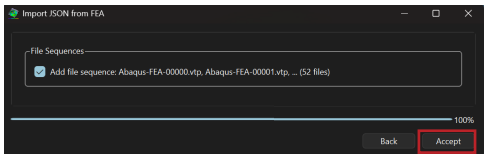


In the top menu bar, click **Extensions** → **Finite element analysis** → **Import JSON from FEA**. Under the **Options** tab in the extension dialog, search for the *.json* file, add a prefix as needed, adjust the scale, and click **Run**. When the conversation is complete, click **Continue**.

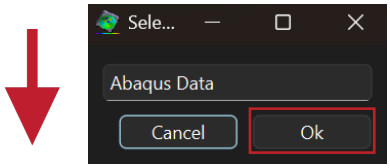




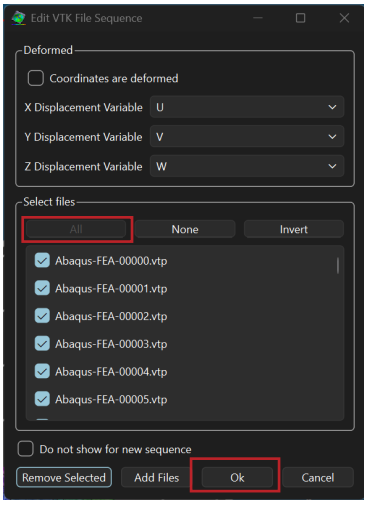
Next, the option is given to automatically import the file sequence after conversion. To do so, make sure the box is checked, then click **Accept**.



Give the sequence a name and click **OK**.



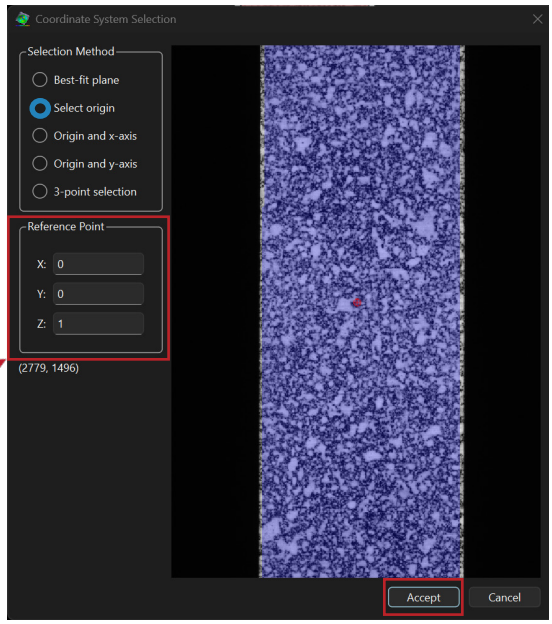
The subsequent window **Edit VTK File Sequence** provides additional options for file importation. For this extension, under the **Select Files** section, click **All** and then click **OK**.



The individual VTP file will be used to import the FE mesh in the upcoming step.

Step 2. Select Coordinate System

To simplify the workflow in the mesh alignment workspace, the coordinate system should be at approximately the correct Z-height. This can be done by clicking **Data** → **Coordinate Tools** → **Select Coordinate System** after performing the initial analysis.



In the Coordinate System Selection dialog, use the 'Select origin' tool to click anywhere on the surface of the specimen to provide a "Reference Point" value that matches the specimen's thickness.

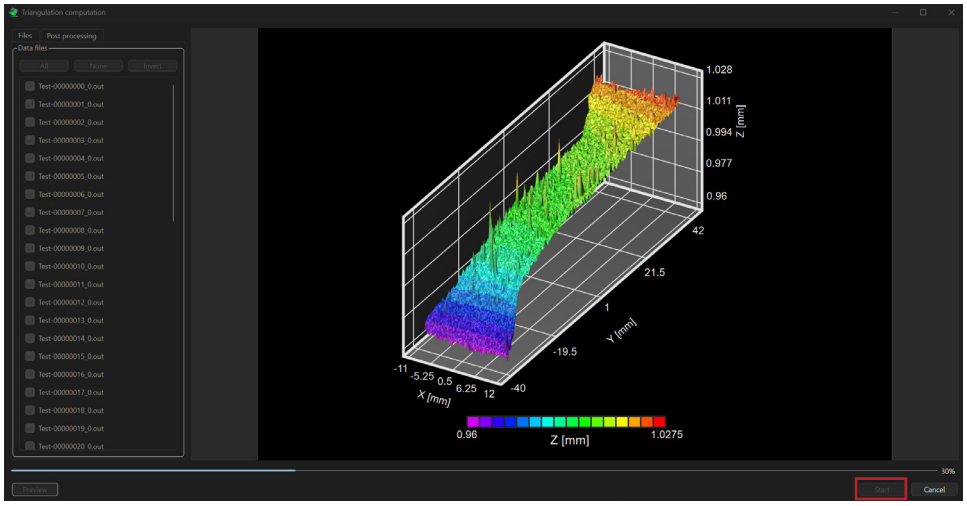
In this example, the specimen is 1mm thick, so a reference point of **0,0,1** is used.

Click **Accept** and then **Set** to apply the transform to the current project.

Anytime you change the coordinate system, remember to retriangulate the dataset. To do this, click on the Restore Data icon in the main tool bar.

Then click **Start**.

When the retriangulation process is complete, click **Close**.

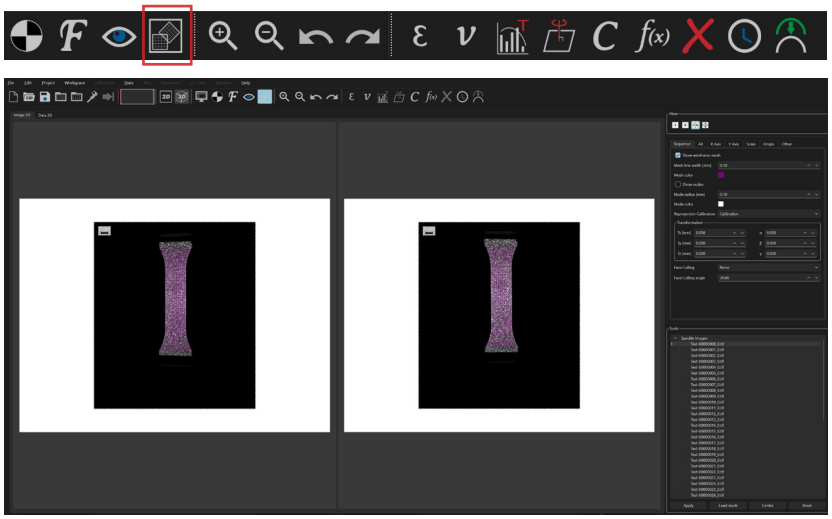


Step 3. Mesh Alignment

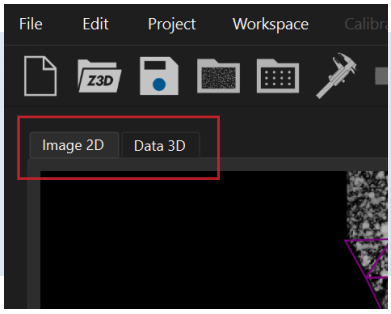
The **Mesh Alignment Workspace** allows VIC-3D to use the calibration information of the camera system to project the FE mesh into each cameras view.

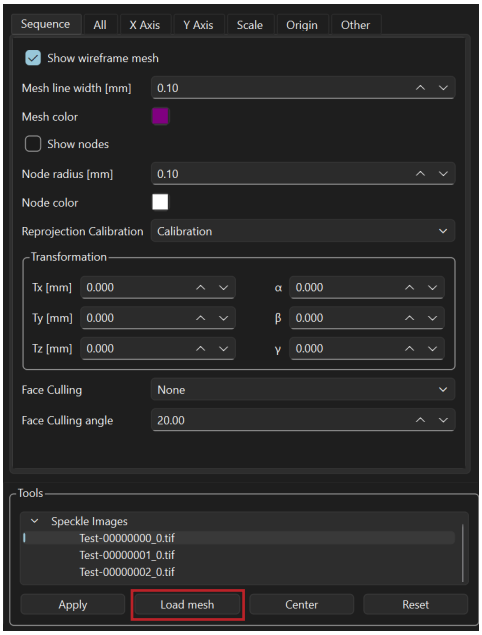
Access the Mesh Alignment Workspace by clicking on the icon in the top icon bar.

In the **View** tab to the top far right, select the side by side view.



The "Engineering" strain in VIC-3D is not the raw displacement gradient `dU/dX`; rather, it is derived from the Lagrange tensor (shown to the right) so that it remains insensitive to rigid-body rotation.



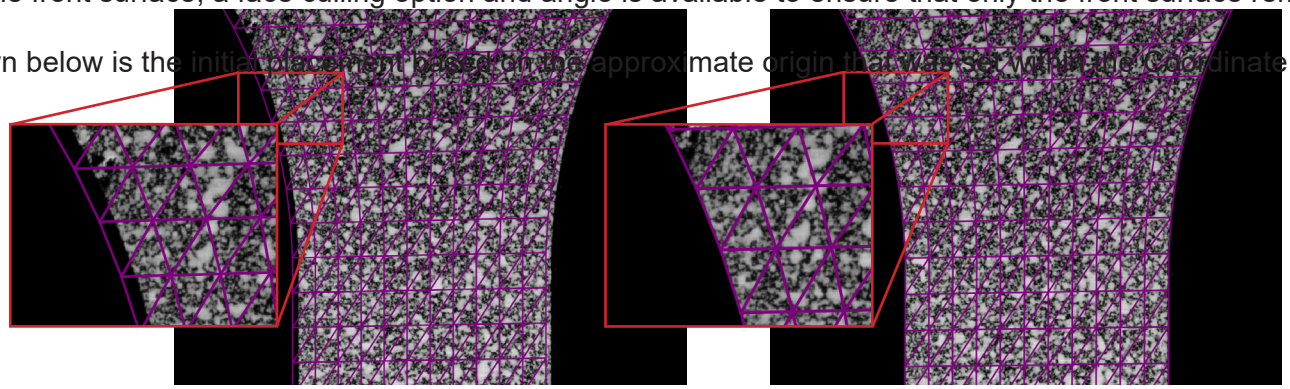


At the bottom of the **Tools** tab to the bottom right, click **Load mesh** to import the FE mesh from the reference unloaded state. Select the .vtp file (*abaqus-FEA-0000.vtp*) created in the previous step and click **Open**.

Visually align the mesh to both images using the transform tools found

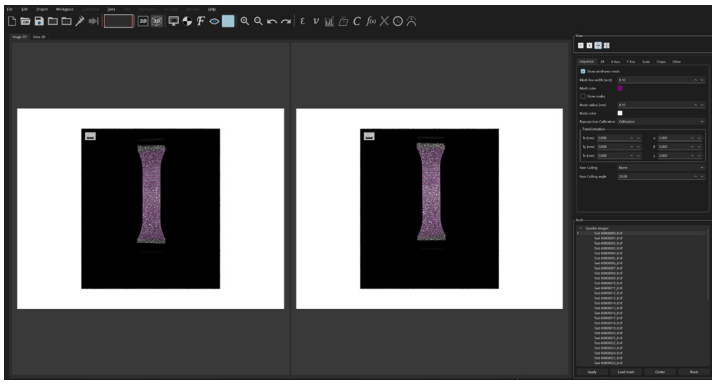
the middle section of the panel on the right (typically this will only require translations in x and y, and rotations in γ because our z was set to the exact value in the previous step). If your FE mesh contains the full model and not just the front surface, a face culling option and angle is available to ensure that only the front surface remains.

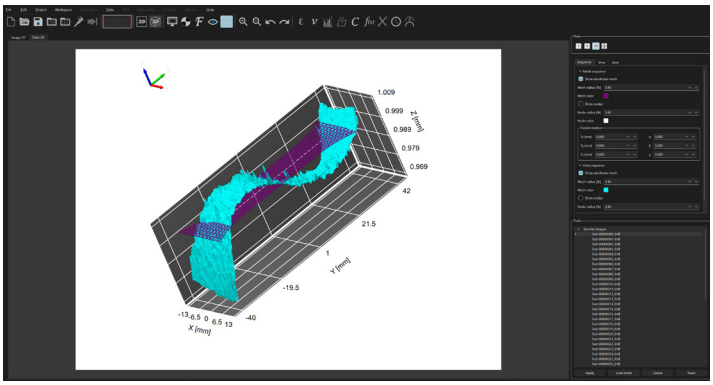
Shown below is the initial placement based on the approximate origin that was set with the Coordinate Tools



Step 4: Confirm Alignment with 3D View

Changing from the 2D view to the 3D view confirms that the alignment is correct.





Finally, click apply at the bottom of the Tools tab to the bottom right, then return to the Main Workspace to retriangulate the data by clicking on the the **Restore Data** icon as before.

Working with Extensions to Produce DIC & FE Results

At this point, all of the data is in a common coordinate system and established within the project. A single extension, **Compare DIC and FE**, performs the like-for-like comparison. It interpolates the FE results onto the DIC measurement points and, when comparing strain, recomputes strain for both fields at those points using the same algorithm VIC-3D applies to the DIC displacement fields. Because one operator and one set of points serve both sides, the resulting difference isolates genuine model–experiment discrepancies from the artifacts that mismatched numerical methods would otherwise introduce. Earlier releases required a sequence of separate extensions to achieve this; the step below replaces that sequence.

> ***PLEASE NOTE:*** *Every extension has a well-documented HELP section within its dialog to explain options and operation.*

Step 5: Compare the DIC and FE Data

There are several comparison options available within the dialog, The choice determines how much of the difference is attributable to the model.

Open the extension by clicking **Extensions** → **Finite element analysis** → **Compare DIC and FE**.

	1 **Solver output as exported**	2 **Displacement only**	3 **Leveled strain**
COMPARE	The solver's variables as it exported them	`U`, `V`, `W`	Strain, recomputed for both fields at the DIC points
SET COMPARE TO	**Direct subtraction - all matching variables**	**Direct subtraction - all matching variables** — the same mode as 1. Read only the displacement differences.	**Strain - recomputed & leveled** — the default.
SETTINGS THAT MATTER	Set **VTK filename** to match the files written by *Import JSON from FEA*. The default is `fe_{IMG_NO:04d}.vtp`.	As 1. No strain setting applies in this mode.	Also set **Strain window size** and **Tensor type**, which apply to both fields in the one run. Defaults are **15** and **Lagrange**.
OUTPUTS	`ΔU ΔV ΔW ΔZ`, plus every solver variable that shares a name with a DIC variable	`ΔU ΔV ΔW`. `ΔZ` is a node-height difference, so it reports geometry rather than displacement.	`Δexx Δeyy Δexy Δe1 Δe2 Δgamma`. Both sides are kept, as `exx_fe` and `exx_dic`.
USE IT FOR	Units, sign, load-step pairing, and the direction of the Z axis. Also any field DIC does not measure, such as stress or temperature.	Stiffness, elastic constants, and boundary conditions.	The quantitative strain result, and the residual FEMU drives to zero.

Under the **Options** tab, set **Compare** to the quantity you chose from the table above.

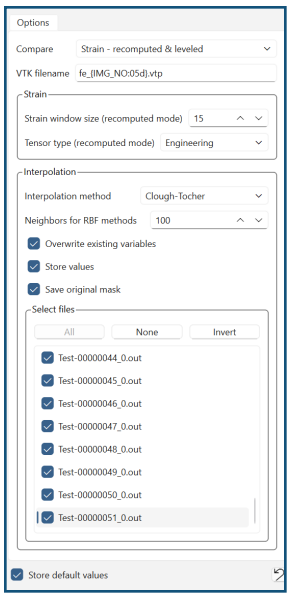
Set **VTK filename** to the template that matches the `.vtp` files written in Step 1.

For **Strain - recomputed & leveled**, set **Strain window size** and **Tensor type**. Both apply to the FE and the DIC field within the same run.

Leave **Interpolation method** on **Clough-Tocher** for a flat specimen. Select **3D surface (barycentric)** for a curved one.

Select the data files to process in the list at the bottom of the dialog.

Click **Run**.



Naming Conventions

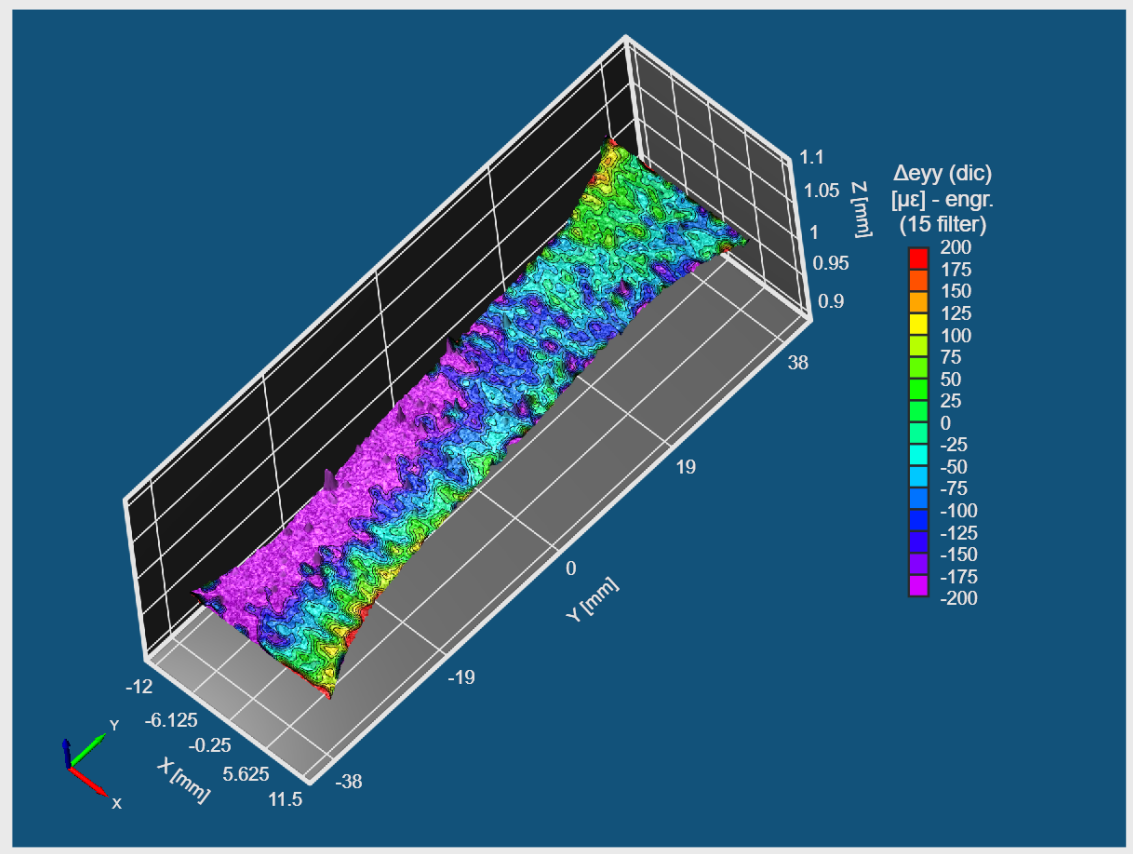
The **VTK filename** *must match the files produced in Step 1, including any prefix added during the conversion. The default template is `fe_{IMG_NO:04d}.vtp`. A template that does not match reports `Could not find FE file` and skips every frame, which looks like a failed import rather than a filename error.*

Every difference is written as `Δ` prefixed to the variable name, in the same units as the source variable. **The sign convention is DIC − FE**, so a positive `Δexx` means the measurement read higher than the model.

Direct subtraction writes `ΔU`, `ΔV`, `ΔW`, `ΔZ`, and a `Δ` variable for every other quantity present in both datasets. `ΔZ` is a difference in node height, so it reports surface geometry rather than displacement.

Strain - recomputed & leveled writes `Δexx`, `Δeyy`, `Δexy`, `Δe1`, `Δe2` and `Δgamma`. Both source fields are retained as `exx_fe` and `exx_dic`, so either side can be contoured on its own.

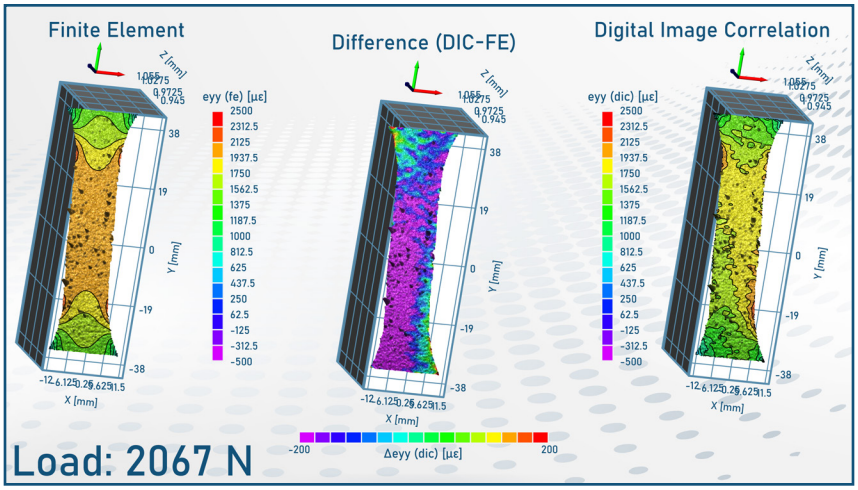
Points that fall outside the FE mesh are masked, and the status line reports the count. It states how much of the measured field entered the comparison.



Visualizing and Reporting within *iris*

Paired DIC and FE contours should be plotted on the same color scale, in the same units, and at the same view angle. Mismatched scales are the single most common reporting error and will visually exaggerate or hide real differences.

DIC, FE, and the different fields are best shown side by side rather than DIC and FE alone, and every figure should include the load level in physical units.



Common Pitfalls

- Auto plane-fit left on — the most common single failure mode in VIC-3D coordinate registration.
- The DIC Z-axis points from the specimen toward the cameras by default; FE models often have Z pointing the opposite way. Mismatch produces sign-flipped W (out-of-plane) displacements.
- Surface offset - DIC reports the painted surface; if paint thickness or curvature offset is non-negligible, the FE surface nodes are at a slightly different location. This is usually negligible, may be a consideration for thin specimens or sharp curvature.

The example data referenced throughout this document was acquired on an aluminum dog-bone specimen at 4 Hz, producing 52 image pairs. The recorded loads from each frame were used as load steps in the FE model, giving a 1:1 correspondence between DIC frames and FE solutions. The steps above begin after initial DIC analysis. The data from this test is available for download. Click the QR code to the right.

Support

If you have any questions, comments, or concerns about using your DIC system, please contact our support team at support@correlatedsolutions.com or 1.803.926.7272.

Click or scan the QR code to the right, or visit correlatedsolutions.com/support, for the knowledge base, software downloads, and trial requests.