

These are copies of the HTML help pages, which are available in Ansys once the tool has been installed successfully. Please notice that the links will not work in this PDF version. However, the content is complete.



Validation Tool: Documentation

This Guide offers you help on how to use the Dantec Dynamics Validation Tool.

Table of Contents

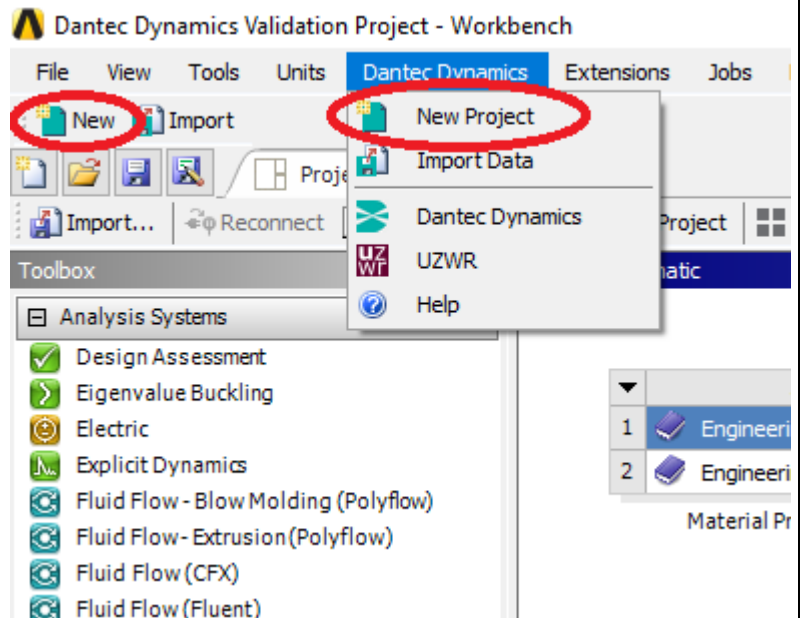
- [1. User's Guide](#)
- [2. HDF5 to TXT conversion Tool](#)
- [3. How to handle contacts in your Model](#)

User's Guide

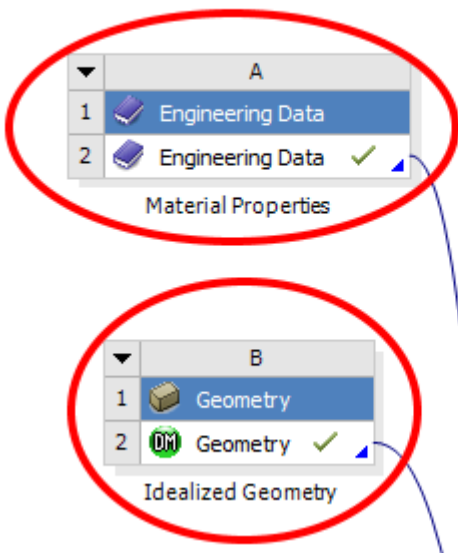
Step 1: Starting a new Validation Project

Start a new validation project by clicking the **New Project** button in the Dantec Dynamics menu or the **New** button on the Dantec Dynamics toolbar.

This will load the Dantec Dynamics Validation Tool template and ask you where to save the new project.



Project Schematic

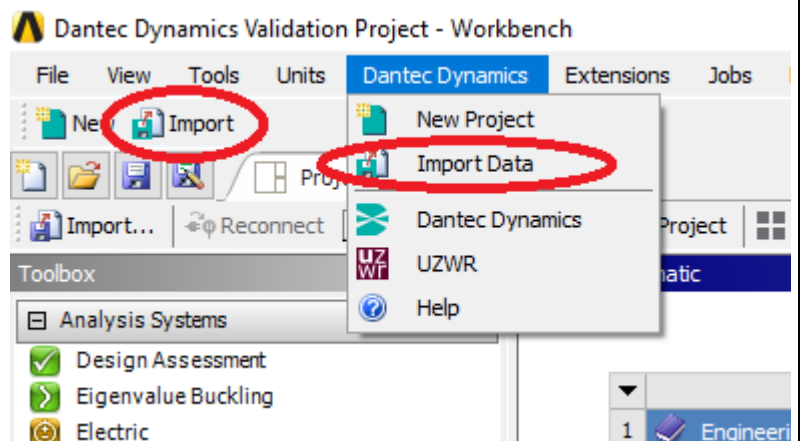


Step 2: Defining your Engineering Data and Geometry

Define your Engineering Data and Geometry as usual.

Step 3: Importing your Dantec Dynamics measurement data

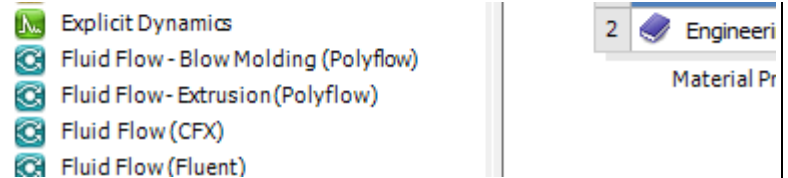
Next import your Dantec Dynamics measurement data, by clicking either **Import Data** in the Dantec Dynamics menu or the **Import** button on the Dantec Dynamics



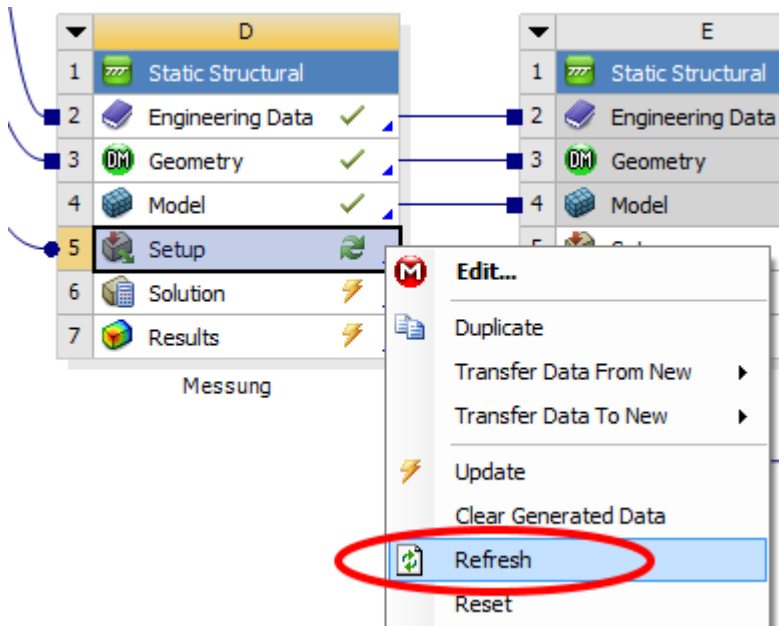
toolbar.

You can choose between importing from a .txt file or .hdf5 file. In the latter case HDF5 data will be

convertet to a .txt file, you will be asked to select a the reference-step .hdf5 file, to select the .txt file with the reference markers (exportet from Istra4D) and to enter the coordinates of the reference markers in your Ansys geometry coordinate-system. See [HDF5 to TXT conversion Tool](#) for more details.



This will update the data in **System C - External Data**.



Step 4: Uploading the Engineering, Geometry and measurement Data

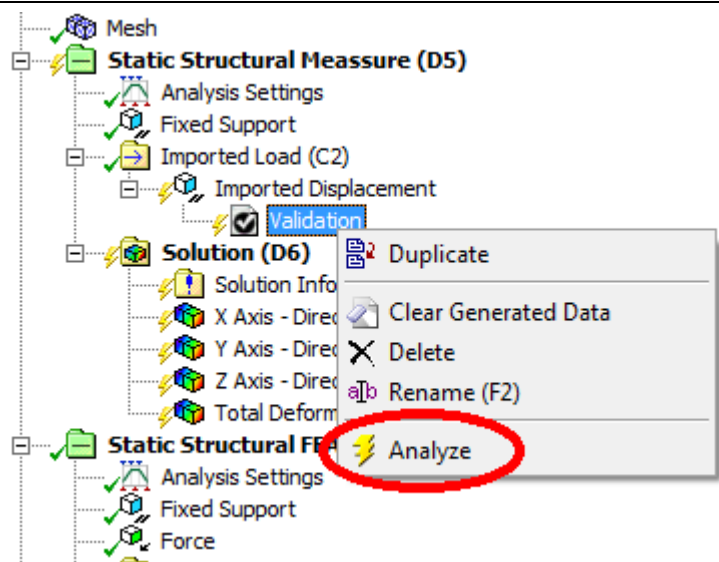
Now you have to re-read the upstream data in the **Static Structural D: Measurement**. You can do this via the right-click context-menu **Refresh**.

Alternatively Ansys will ask you to re-read the upstream data, when you open the **Ansys Mechanical Module**, which you will have to do for the next step. Simply click on **yes**

when you are asked.

Step 5: Mapping the Measurement data on your Geometry

Start the **Ansys Mechanical Module**. Under the **Static Structural Measure** entry in the Project tree, insert the proper boundary conditions for your model and under the subentry **Importet Load** right-click on **Validation** and choose **Analyze** in order to print your measurement data on your Geometry.



Now you can check the mapping of your measurement data, **solve** the **Static Sturctural Measurement** system and watch the results of it.

Step 6: Setting up your Simulation Model

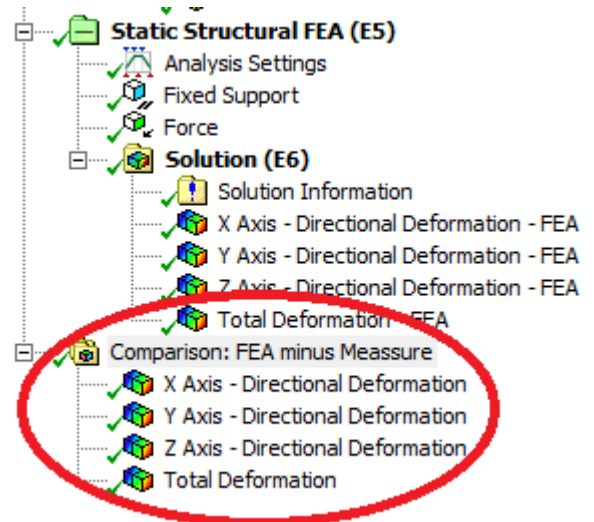
Setup your Simulation Model under **Static Structural FEA**.

If you have **contacts** in your Simulation Model, read [How to handle contacts in your Simulation Model](#).

Step 7: Comparing the results

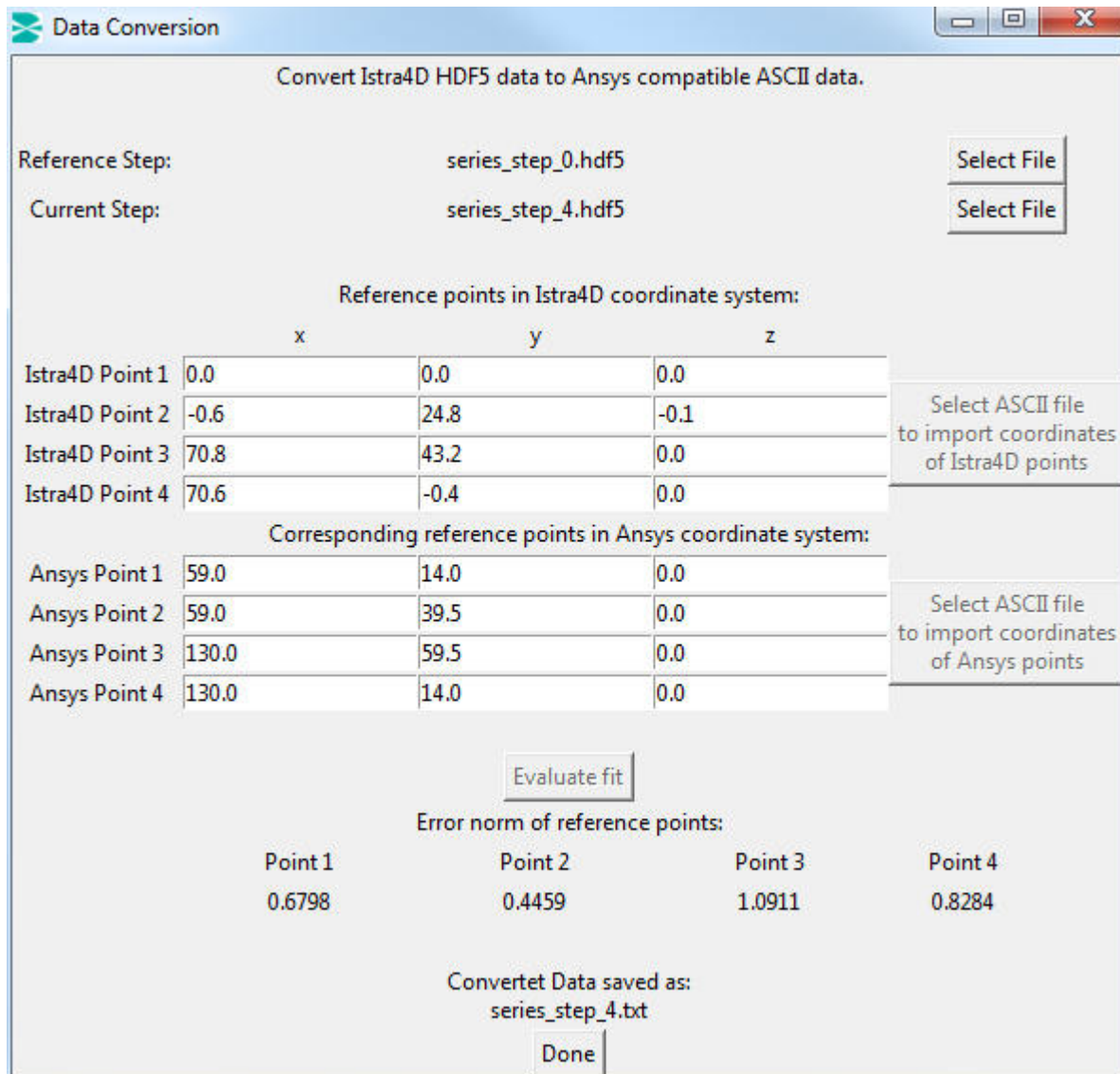
You can view the difference between the mapped measurement results and the simulation results in the **Ansys Mechanical Module**. In order to do so, there is an entry **Comparison: FEA minus Measure** in the Project tree.

Note: *As the CFD-Post Module is no longer compatible to the Mechanical Module in recent Ansys Versions, the separate Results Extension has been removed and all results can be viewed as described above.*



HDF5 to TXT conversion tool

1. The Data Conversion window



The image shows the **Data**

Conversion window filled with the data used for the template project, which is created upon **New Project**.

In order to import the measurement data from the Istra4D HDF5 snapshot into ANSYS Workbench, you first need to convert it into an ASCII text file. This is done by this HDF5 to TXT conversion Tool, which is a Python program that is called by clicking **Import Data** in the Dantec Dynamics menu or the **Import** button on the Dantec Dynamics toolbar.

When doing so, the **Data Conversion** window will show up. You then have to select the .hdf5 step files of the **Reference Step** of your measurement, which corresponds to your undeformed ANSYS geometry, and of the **Current Step**, which is the step you want to validate.

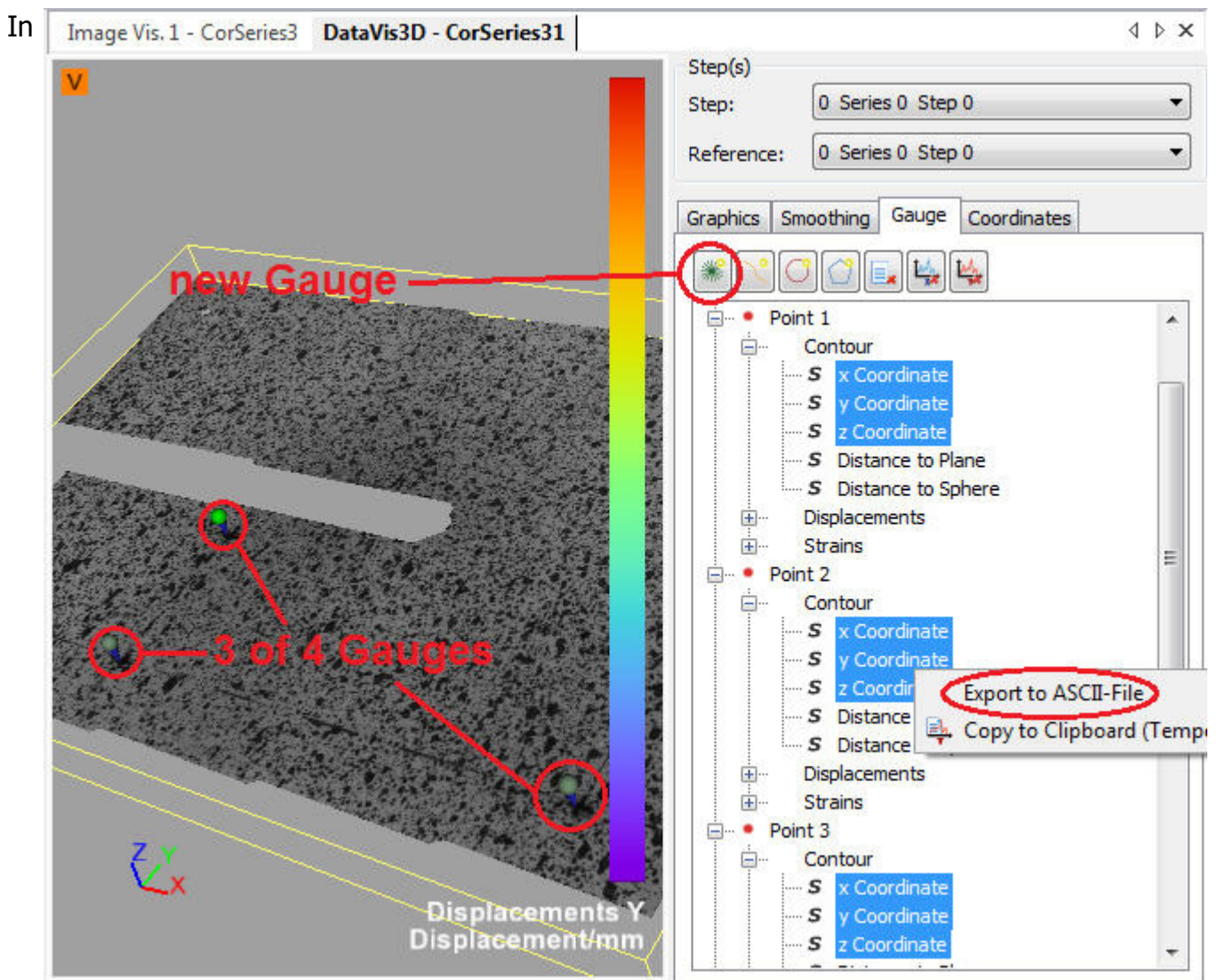
Now you have to enter the coordinates of the four points, which will be used by the program, to map the measurement data on the ANSYS geometry. The coordinates of the reference points in the Istra4D coordinate system can be [exported as ASCII file from Istra4D](#), click on the **Select ASCII file to import coordinates of Istra4D points** button and choose the appropriate file. The coordinates will show up on the table in the conversion tool window. Alternatively you can input them manually.

The coordinates of the reference points in the ANSYS coordinate system can be imported similar to the Istra4D points. See the [format notes](#) for importing them from an ASCII file.

Next click on **Evaluate fit** to let the program calculate the mapping of the Istra4D measurement data on the ANSYS geometry and generate the according ASCII file. The conversion tool will also display the error norm between the ANSYS reference points and the mapped Istra4D reference points.

The converted data is saved as a .txt file under the same directory and with the same name as the chosen **Current Step** file. Furthermore the file will be automatically updated in the ANSYS Workbench Project.

2. Istra4D - Export Gauge coordinates as ASCII file



Istra4D, you first have to set 4 Gauges at the (somehow marked) reference points on the object. In the example on the right, we marked these points with painted crosses on the

measurement surface.

Then select the x-, y- and z-coordinates (under the **Contour** entry) of all 4 points, Right-Click on one of them and select **Export to ASCII-File**.

In the upcoming dialog, choose where you want to save the exported data and click on **Next >**. In the **Step selection** dialog choose the step, which you want to use as your reference step in the ANSYS simulation.

3. Format for ANSYS coordinates in ASCII file

If you want to import the coordinates of the reference points in the ANSYS coordinate system via an ASCII file, simply write them semicolon sperated in one line in a text file. For the example in the picture above, this would look like this:

```
59; 14; 0; 59; 39,5; 0; 130; 59,5; 0; 130; 14; 0
```

It doesn't matter if you use colons or dots as decimal mark.

How to handle contacts in your Simulation Model

In case you have to handle contacts in your Simulation Model, the template project loaded by starting a **New Project** is not sufficient. The reason for this is, that contacts are part of the **Model**-level, so if you add contacts in the template project they will be taken into account for the measurement as well.

To avoid this you have to undock the **Model** between the **Measurement**- and the **FEA-System** and set them up more independently, in such way, that only the **Engineering Data** and **Geometry** might be combined. Doing so will result in you not being able to compare the **Measurement** and the **FEA** results directly in the **ANSYS Mechanical Module**. In order to be able to make this comparison again, we suggest to add a additional **Static Structural** System which includes the results (as shown in the picture below) and handles the comparison.

