WE MechLoad Viewer

The Complete User Manual

Chapter 6: Exporting for Ansys

One of the most powerful features of the WE MechLoad Viewer is its ability to directly generate input files for use in **Ansys Mechanical**. This feature streamlines the process of taking processed test or simulation data and applying it as a load in a subsequent Finite Element Analysis (FEA).

The export functionality is located in the **Part Loads** tab.

6.1 The Export Workflow

The process is designed to be straightforward. You use the controls on the Part Loads tab to isolate and process the exact data you need, and then you export it.

Step-by-Step Guide to Exporting for Ansys:

1. **Navigate to the Part Loads tab.**
2. **Filter Your Data:** Use the Side Filter Selector and Component Filter Selector to narrow down the data to the specific loads you want to export. For example, you might filter for "Side" = EngineMount to select all engine mount loads.
3. **(Optional but Recommended) Process Your Data:**
   - For **Time-Domain Data**, you can use the Apply Data Section to export only a specific time interval of interest.
   - You can also Apply Tukey Window to properly condition the data for frequency analysis within Ansys (e.g., for a Response Spectrum analysis).
4. **Click the Export to Ansys Button.**
5. **Select Parts to Export:** A new dialog window will appear, listing all the "Sides" available in your data. This is your final chance to confirm which parts you want to include in the export. You can select multiple items from this list (e.g., by holding Ctrl while clicking). Click **Confirm**.

What Happens Next? The Output Files

After you confirm, the application will work in the background and save several files in the same directory where the application is running. These files are the key to your Ansys simulation:

- **extracted_..._in_original_units.csv:** A CSV file for each part you selected, containing the processed data in its original units. This is for your records.
- **extracted_..._multiplied_by_1000.csv:** Another set of CSV files. Importantly, the load values in these files have been multiplied by 1000. This is to handle a common unit

conversion (e.g., from kN to N or from MPa to Pa).

- **A .dat file:** This is the main data file for Ansys. It's a specially formatted text file containing the load tables (either load vs. time or load & phase vs. frequency).
- **An .inp file (APDL script):** This is the script file you'll use in Ansys. It contains the APDL (Ansys Parametric Design Language) commands that tell Ansys how to read the .dat file and apply the loads correctly.

6.2 Using the Exported Files in Ansys

The specific steps in Ansys will vary slightly depending on whether you are running a **Transient Structural** (time-domain) or **Harmonic Response** (frequency-domain) analysis.

General Steps in Ansys Mechanical:

1. Set up your FEA model as usual (Geometry, Materials, Meshing, etc.).
2. In the project tree, insert a **Commands (APDL)** object.
3. In the details of the Commands object, copy and paste the contents of the **.inp** file that the viewer generated.
4. **Crucially, you must make sure the .dat file is in a location that Ansys can access**, typically the same folder as your Ansys project file. You may need to edit the file path inside the APDL script if you move the file.
5. Solve the analysis. The commands will automatically create the necessary load tables and apply them to your model.

This feature can save a significant amount of time by automating the tedious and error-prone process of manually creating load tables in Ansys.

Chapter 7: Global Settings and Tips

The **Settings** tab allows you to customize the appearance of all the plots in the application to suit your preferences. Changes made here will apply globally to all tabs.

7.1 Customizing Your Plots

- **Show Legend:** Toggles the plot legend on or off.
- **Show Grid:** Toggles the background grid lines on or off for easier reading of values.
- **Show Rolling Min/Max (Time Data Only):** When checked, this will draw additional lines on your time-domain plots showing the rolling minimum and maximum values of the signal, which can be useful for identifying peaks and troughs.
- **Plot Theme Selector:** Change the overall color scheme of your plots. Choose from several professional themes like plotly_dark or ggplot2.
- **Legend Position:** Control where the legend appears on the plot (e.g., top right, bottom left).

7.2 Keyboard Shortcuts

For quicker access to common settings, you can use these keyboard shortcuts anywhere in

the application:

- **K key:** Cycles through the different legend positions.
- **L key:** Toggles the legend's visibility on and off.

7.3 Troubleshooting Common Issues

- **Issue:** The application doesn't load my data and shows an error.
  - **Solution:** The most common reason is that your data folder is missing either the full.pld or max.pld file, or they are not formatted correctly as pipe-delimited (|) text files. Double-check your input files.
- **Issue:** I can't see the "Time Domain Rep." tab.
  - **Solution:** That's normal! This tab only appears when you have loaded frequency-domain data.
- **Issue:** I checked a box in the Directory Tree, but the plot didn't update.
  - **Solution:** Make sure the dataset you are trying to load has the same domain (Time or Frequency) as the primary dataset. The application will not mix different data types on the same plot.

End of Manual

We hope this manual helps you get the most out of the WE MechLoad Viewer. Happy analyzing!