



Altair

---

**HyperWorks**

Altair MotionView 2019 Tutorials

MV-3030: Load Export

[altairhyperworks.com](http://altairhyperworks.com)

## MV-3030: Load Export

The **Load Export** utility allows you to bridge the gap between Multi-Body Dynamics (MBD) analysis and Finite Element (FE) analysis using MotionView by:

- Identifying and summarizing all loads acting on one/multiple body(ies) for any given time step(s) in a tabular format.
- Identifying and transferring all the forces and moments for one component at any given time step(s) to a NASTRAN input deck that contains GRID, CORD, FORCE, and MOMENT cards.

### Using Load Export

To use this utility, specify the components in the MotionView model for which loads are to be processed. You can do this by:

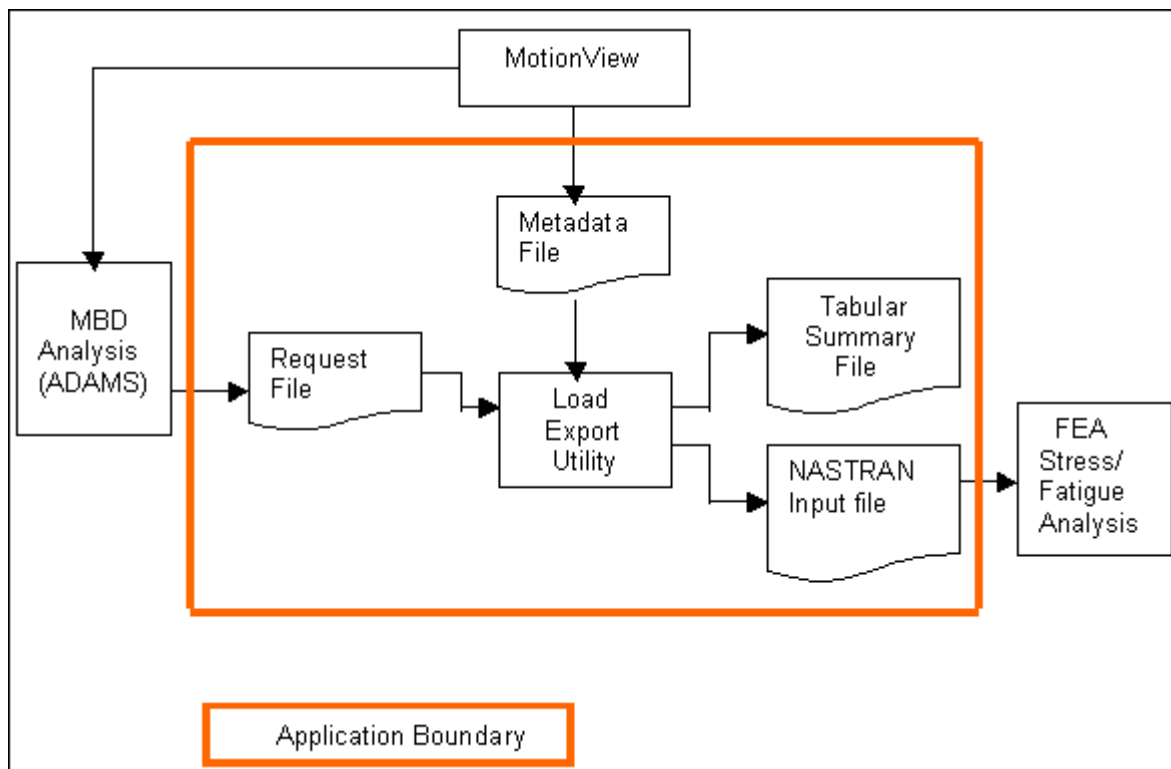
- Using the MotionView Interface.

OR

- Editing the MDL model file to add force output requests on body(ies).


When performing the MS/ADAMS solver run on the MotionView model, you will get a **metadata** file (an ASCII file written out from MotionView that contains information about force output on a body).

This file along with the solver output files viz. MS (\*.plt) or ADAMS (\*.req) become the input files for this utility. The application scope of this utility is shown in the figure below:



Interaction diagram

## Step 1: Creating a Metadata File and Launching Load Export.

1. Copy the `load_export.mdl` file, located in the `mbd_modeling\externalcodes` folder, to your <working directory>.
2. Start a new MotionView session.
3. Load the front vehicle model file `load_export.mdl`, located in <working directory>.
4. Right-click on **The Model** in the **Project Browser** and select **Add General MDL Entity > Output**, or right-click the **Outputs** icon, , on the **Model-General** toolbar.


The **Add Output** dialog is displayed.

5. Accept the default selections and click **OK**.
6. Use the drop-down menu to change the **Output** type from the default **Displacement** to **Force**.
7. Double-click the **Body** collector.

The **Select a Body** dialog is displayed.

8. Expand the model-tree.
9. In the **Frnt macpherson susp** system folder, expand the **Bodies** folder and select the body **Lwr control arm – left**. (or you can pick the **Lwr Control arm - left** directly from the model in the graphics area by clicking the **Body** collector once).

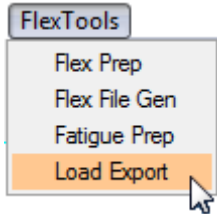


10. Repeat steps 4 through 9 to create an output force request on **Lwr control arm – right**.
11. Click the **Run Solver** icon .
12. From the **Main** tab, change **End Time** to 2 seconds.
13. Save the solver input file as `load_export.xml`, to the <working directory>.
14. Click on the **Run** button, to solve the model in **MotionSolve**.

MotionView creates a metadata file named `load_export.meta` in the <working directory>.

## Step 2: Using the Load Export Utility and Generating a NASTRAN Input Deck.

1. From the **Flex Tools** menu, select the **Load Export** utility.



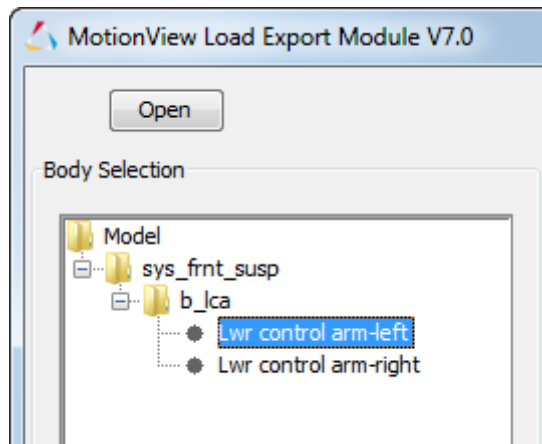
Launching the Load Export utility



The Load Export utility

- From the **Load Export** panel, open the file `load_export.meta`, located in <working directory>.

All bodies for which force outputs are requested are displayed in a tree structure in the **Body Selection** panel. You can select one or multiple bodies from the tree. In this step select the body **Lwr control arm-left**.

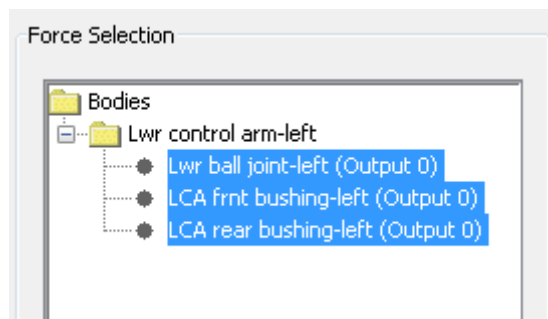


Body Selection panel

- Expand the **sys\_frt\_susp** folder and select the body **Lwr control arm – left**.

All the forces acting on the **lwr control arm – left** are displayed in the **Force Selection** panel. You can choose any number of loads acting on the body. Only the loads selected by you are exported by the utility.

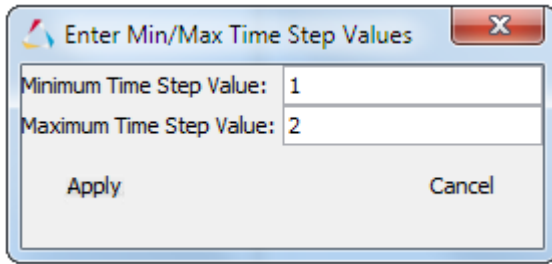
- Select all three forces acting on **Lwr control arm – left**.



Force Selection panel

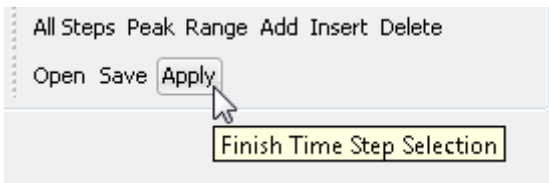
- The **Time Selection** panel allows you to enter/select the time steps for which the loads are to be exported.
- Click the **Range** button.

7. The current simulation runs from 0 to 2 seconds. Specify a **Minimum Time Step Value** of 1 and a **Maximum Time Step Value** of 2.



Activating the Export panel

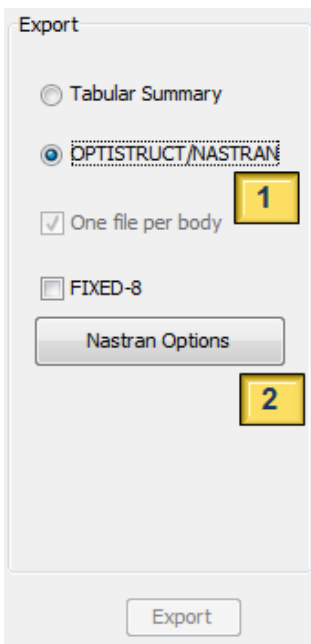
8. Click **Apply**.
9. Enter Min/Max Time Step Values.
10. Click **Apply** on the **Time Selection** panel.



This activates the **Export** panel.

**Note** After time step input, you must click the **Apply** button to verify the validity of the time steps. If a time step entered is not present in the ADAMS request file, an error message is generated and you must make appropriate corrections.

11. Select **OPTISTRUCT/NASTRAN** [1] by using the radio button under the **Export** panel.



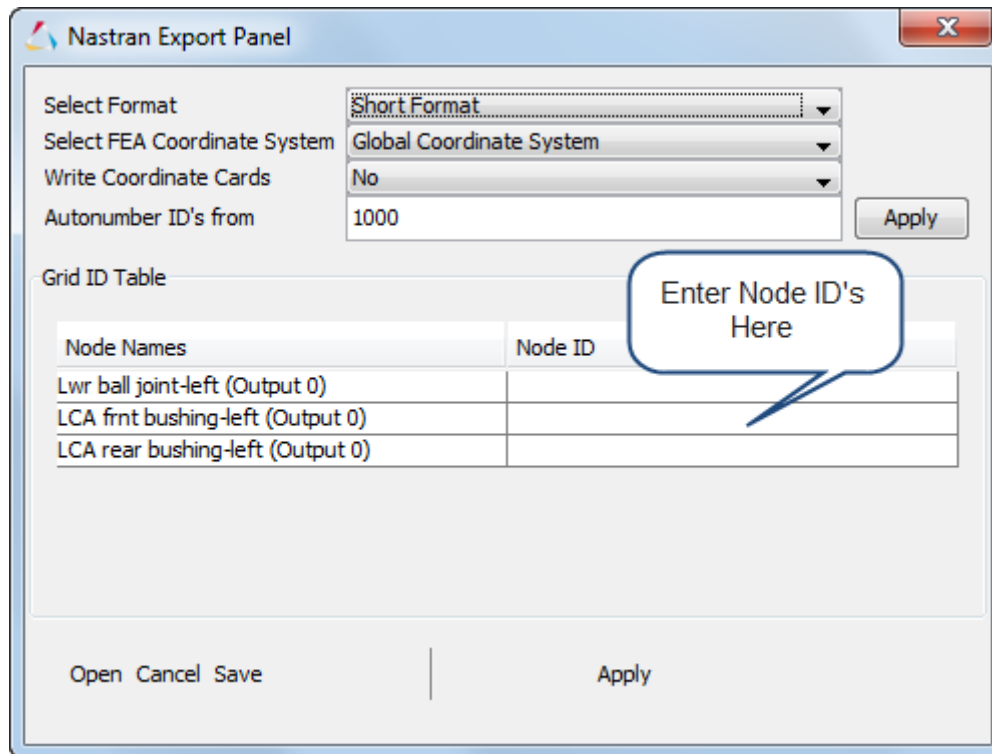
Nastran options

12. Click **Nastran Options** [2] to launch the **Nastran Export Panel**.

This dialog allows you to enter the **Nastran node ID** numbers in the second column of the table.

You can specify three additional options:

- the Nastran deck format (**Large/Small**)
- the reference frame (**LPRF/Global**) in which the GRID cards are written
- whether or not to explicitly output the CORD1R card in the Nastran input deck (**Yes/No**)



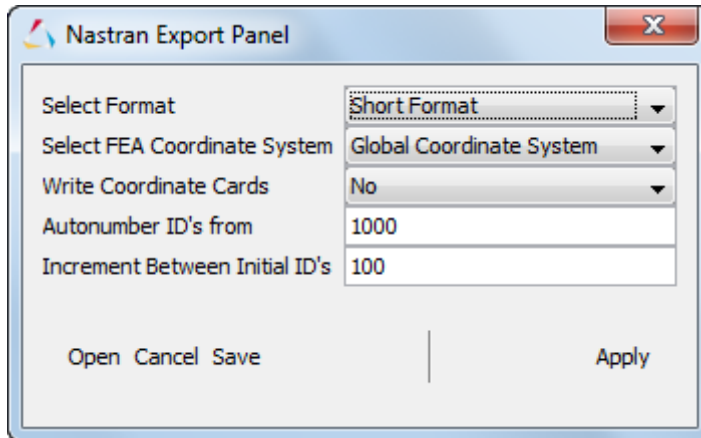
13. Accept the default selections in the **Nastran Export** dialog.
14. Specify the **Node ID's** as follows:
- Lwr ball joint – 1
  - LCA rear bush – 2
  - LCA frnt bush – 3
15. Click **Apply**.
16. Click **Export** on the **Load Export** panel.
17. Specify a filename.

18. Click **Save**.

This creates a *subcase* file, in addition to the Nastran input deck, in the same directory as the *.dat* file.

19. Repeat steps 3 through 18 to export the loads on the **Lwr control arm – right**.

**Note** In point 2 above, if you select multiple bodies, the **Nastran Export Panel** will look as shown below:



Nastran Export Panel for multiple body selection