Adams 2023.1

Getting Started: Adams Car Ride

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
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/

Disclaimer

ADAM:V2023.1:Z:Z:Z:DC-GS-RIDE

Documentation Feedback

At Hexagon Manufacturing Intelligence, 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 Manufacturing Intelligence 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.

Getting Started Using Adams Car Ride

Introducing Adams Car Ride

Overview

This chapter introduces you to Adams Car Ride. It contains the following sections:

- About the Tutorials
- Starting Adams Car Ride
- Getting Help Online

About the Tutorials

We assume that you will work through the tutorials in sequential order. Therefore, we give you more guidance in the beginning and less as you proceed through each tutorial.

Note:	If you choose not to work through the tutorials in sequential order, you may have to reference earlier tutorials for some of the basic concepts.

- Component Template Tutorial - under development.
- Four-Post Test Rig Tutorial - Teaches you how to perform full-vehicle analyses and analyze the results of the analyses.

Starting Adams Car Ride

Because Adams Car Ride is a plugin to Adams Car, you first start Adams Car and then load Adams Car Ride.

In the Linux environment, you start Adams Car from the Adams Toolbar. For information on the Adams Toolbar, see the guide, Running and Configuring Adams on **Linux**.

In the Windows environment, you start Adams Car from the **Start** button. For more information, see the guide, Running and Configuring Adams on **Windows**.

To load Adams Car Ride:

1. Start Adams Car as explained in Starting Adams Car Standard Interface in the guide Getting Started Using Adams Car.
2. From the **Tools** menu, select **Plugin Manager**. For help on the Plugin Manager, press **F1** when the dialog box is active.
3. In the list of plugin names, find **Adams Car Ride**, and then select one or both of the following:
   - **Load** - Loads Adams Car Ride in the current session.
   - **Load at Startup** - Instructs Adams Car to load Adams Car Ride in all future Adams Car sessions.
4. Select **OK**.

   Adams Car loads Adams Car Ride. The interface now includes a new menu, Ride.

Getting Help Online

When working in Adams Car Ride, you can get help as follows:

- From the **Help** menu, select **Adams Car Ride Help**.
- While working in any dialog box, press the **F1** key.

Four-Post Test Rig Tutorial

Overview

This tutorial guides you through the process of running several analyses on a full vehicle and observing the results of the analyses.

This chapter includes the following sections:

- Running a Swept-Sine Analysis
- Simulating a Rough Road
- Simulating a Single Bump

This tutorial takes about two hours to complete.

Starting Adams Car Ride

Because Adams Car Ride is a plugin to Adams Car, you first start Adams Car and then load Adams Car Ride.

In the Linux environment, you start Adams Car from the Adams Toolbar. In the Windows environment, you start Adams Car from the **Start** button. For more information, see the online help for Running and Configuring Adams.

To load Adams Car Ride:

1. Start Adams Car as explained in Starting Adams Car Standard Interface in the guide Getting Started Using Adams Car.
2. From the **Tools** menu, select **Plugin Manager**. For help on the Plugin Manager, press **F1** when the dialog box is active.
3. In the list of plugin names, find **Adams Car Ride**, and then select one or both of the following:
   - **Load** - Loads Adams Car Ride in the current session.
   - **Load at Startup** - Instructs Adams Car to load Adams Car Ride in all future Adams Car sessions.
4. Select **OK**.

   Adams Car loads Adams Car Ride. The interface now includes a new menu, Ride.

Running a Swept-Sine Analysis

In this section, you perform a simulation using a basic ride input: a vertical swept-sine displacement disturbance of the tires.

Opening the Assembly

You first open the assembly on which you will perform the analysis.

To open the assembly:

1. From the **File** menu, point to **Open**, and then select **Assembly**.
2. Right-click the **Assembly Name** text box, point to **Search**, and then select **<aride_shared>/assemblies.tbl**.
3. Double-click **Vehicle_full_4post_PAC2002.asy**.
4. Select **OK**.

Setting up the Analysis

Now that the assembly is open, you are ready to set up the analysis.

To set up the analysis:

1. From the **Ride** menu, point to **Full-Vehicle Analysis**, and then select **Four-Post Test Rig**.

   The Full-Vehicle Assembly text box displays the name of the assembly, `Vehicle_full_4post_PAC2002`.

2. Enter the following specifications:
   - **Output Prefix:swept_heave**
   - **End Time:10**
   - **Mode of Simulation:interactive**
   - **Basis for Number of Output Steps:min. number of outputs per input**
   - **Target Value for Basis:20**

     To prevent errors caused by aliasing, sample outputs at a minimum of six times the highest input frequency. A ratio of ten is much better than six; 20 is very good. Never use a ratio less than six.

     Adams Car Ride automatically fills in the values under Actual Values Used for Simulation.
   - **Actuation Type:displacement**
   - **Input Source:swept sine**
   - **Input Locations:beneath tires**
   - **Vehicle Constraint: no constraint**
   - **Start Frequency:0.0**
   - **End Frequency:20**
   - **Displacement Amplitude:10 mm**
   - **Excitation Mode:heave**
   - **Active Actuators:all**

3. Select **Apply**.

   The simulation will take a few minutes. Each pad under the tires gets the same displacement: a sine wave of 10 mm amplitude that varies linearly from 0 to 20 Hz in frequency over the 10-second simulation time.

4. When the simulation finishes, close the Message Window.

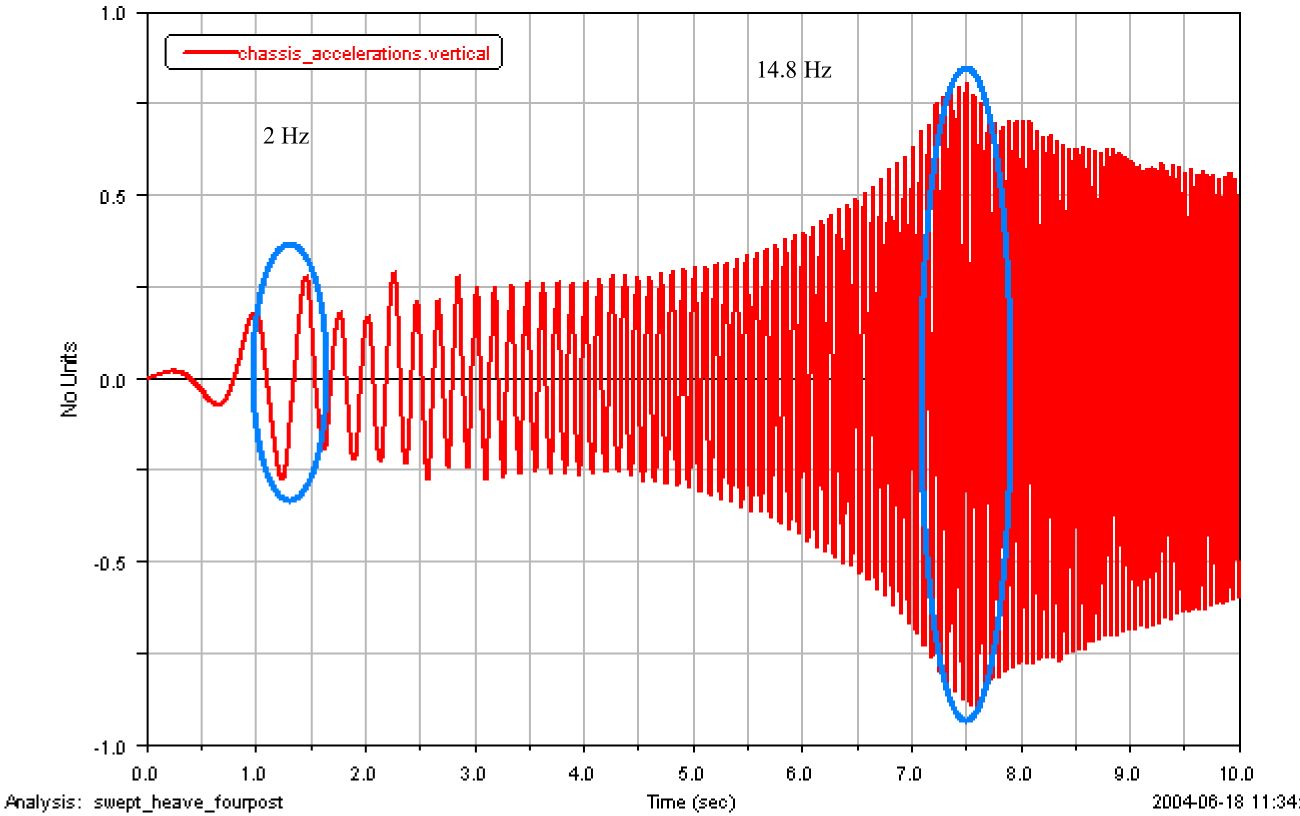
Viewing the Analysis Results

You can now view the analysis results in Adams PostProcessor.

To view the analysis results:

1. From the **Review** menu, select **Postprocessing Window**, or press **F8**.
2. In the dashboard, set **Source** to **Result Sets**.
3. From the **Result Set** list, select **chassis_accelerations**.
4. From the **Component** list, select **vertical**.
5. Select **Add Curves**.

   Adams PostProcessor displays the plot as shown next (we've added the ovals to point out the peaks):



Note the slight peaks in the vertical acceleration amplitudes at about 1.5 seconds and 7.5 seconds. The peaks correspond to input frequencies at about 2 Hz and 14.8 Hz, and show the resonance of the vehicle for the body and suspensions, respectively.

6. To return to Adams Car Ride, press **F8**.

Simulating a Rough Road

In this section, you set up the four-post test rig to accept inputs stored in RPC III file format. You also generate a road profile using a mathematical model, and store it in an RPC III file. Then, you use the road profile you generated to stimulate the vehicle using the four-post test rig.

To learn about RPC III files, see the online help for Adams Durability.

Creating a Road-Profile Data File

To create a road-profile data file:

1. After you've returned to Adams Car Ride, ensure that the **Full-Vehicle Analysis: Four Poster Testrig** dialog box is still open. If it is not, open it as you did in Setting up the Analysis.
2. Specify the following:
   - **Output Prefix: rprof**
   - **End Time: 5**
   - **Basis for Number of Output Steps: output frequency**
   - **Target Value For Basis: 20 Hz**
   - Adams Car Ride automatically fills in the values under Actual Values Used for Simulation.
   - **Actuation Type: displacement**
   - **Input Source: road profiles**
   - **Input Locations: beneath tires**
   - **Vehicle Constraint: no constraint**
3. Select **Set Up Road Profiles**.

   Adams Car Ride opens the Road-Profile Setup: Four Poster Testrig dialog box.
4. Set **Profile Source** to **RPC files**.
5. Set **Vehicle Speed** to **100 km/h**.

   Adams Car Ride displays the time lag. This is the time that the disturbances for the rear wheels lag behind those of the front.
6. Right-click the **File Name** text box for either wheeltrack, point to **Search**, and then select **<aride_shared>/road_profiles.tbl**.

   The Select File dialog box displays two files:
   - `example.rsp` - File created with the Adams Car Ride Road Profile Generation tool.
   - `flat.rsp` - Represents a road with zero height. You could use this file if you want one side of the car to not be excited by an RPC input.
   - For this tutorial, we recommend that you make your own road-profile data file using the Road-Profile Generation tool.
7. Select **Cancel** to close the **Select File** dialog box. You can leave the other dialog boxes open.
8. From the **Ride** menu, point to **Tools**, and then select **Road-Profile Generation**.
9. Enter the following specifications:
   - **Elevation PSD Parameter: Ge:0.1**
   - **Velocity PSD Parameter: Gs:20**
   - **Acceleration PSD Parameter: Ga:0.1**
10. Accept the default values for **Profile Length**, **Sample Interval**, and **Correlation Baselength**.

11. Right-click the **Output Filename for RPC III File** text box, point to **Search**, and then select your **private** database.
12. Double-click the folder **road_profiles.tbl**.
13. In the **File name** text box, enter **tut_road.rsp**.
14. Select **Open**.

    In the Output Filename for RPC III File text box, Adams Car Ride displays the full path to the file you just created: `mdids://private/road_profiles.tbl/tut_road.rsp`.
15. Select **OK**.

Note:	When creating a road profile as described above, the RPC III file will contain road height as function of travelled distance. When setting the 'Input per wheel' checkbox, an RPC III file can be specified for each wheel. However, this RPC III file is supposed to have road height as function of time.

Setting up the Test Rig and Analysis

To set up the test rig and the analysis:

1. Return to the **Road-Profile Setup** dialog box.
2. Right-click the **File Name** text box for either wheeltrack and search your private database for the file you just created, **tut_road.rsp**
3. Repeat the previous step for the other wheeltrack.
4. For the **Left Wheeltrack Profile**, set **Channel Number** to **1**.

Note:	The Road-Profile Generation tool always makes a file with two channels. You could use either one for the left or right wheeltrack, but, by convention, channels 1 and 2 are labeled LElev and RElev, for left and right elevation, respectively. To obtain symmetrical input, you can use the same channel number for both wheeltracks.

5. For the **Right Wheeltrack Profile**, set **Channel Number** to **2**.
6. Select **OK**.

   The four-post test rig is now set up for the road profile you just created.
7. On the **Full-Vehicle Analysis** dialog box, select **Apply**.

   The simulation will take a few minutes.

Viewing the Analysis Results

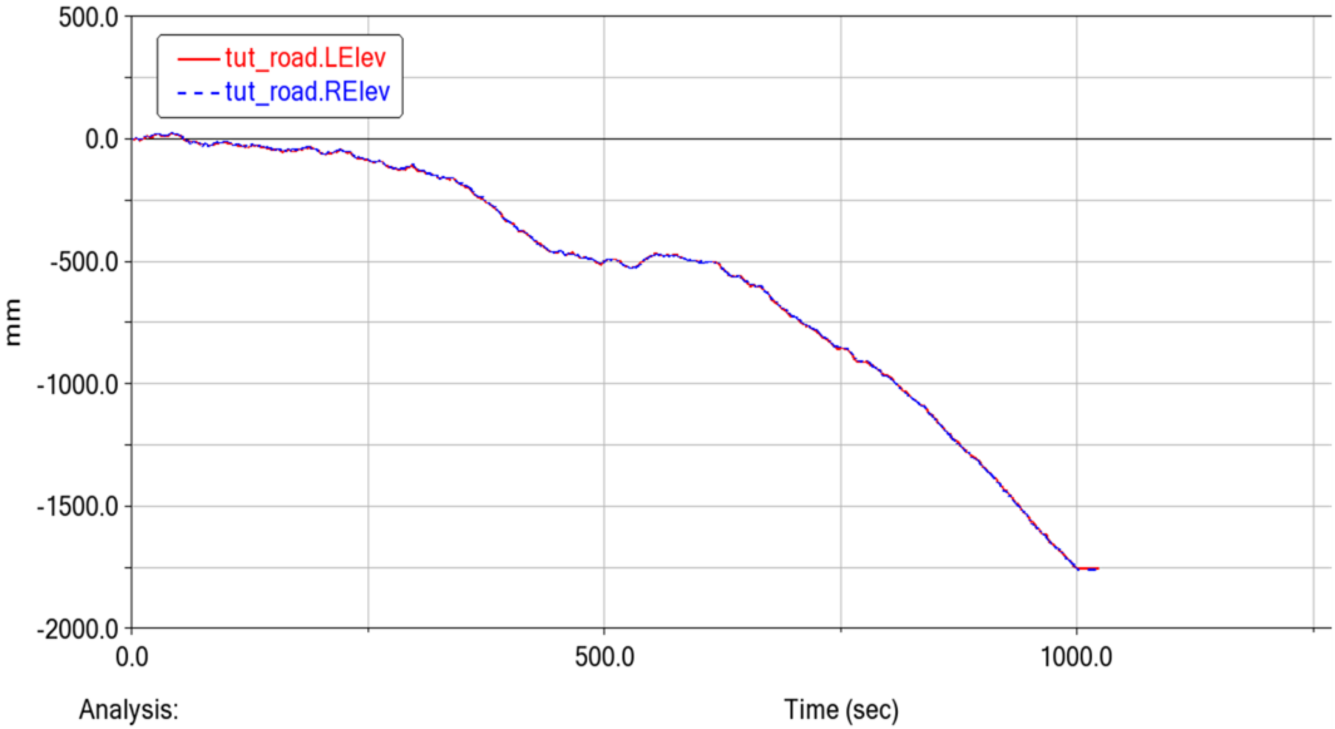
You can now view the analysis results in Adams PostProcessor. First you will view an animation and then you will look at the profile that causes the vehicle behavior you saw in the animation.

To view the analysis results:

1. Start Adams PostProcessor just as you did in Viewing the Analysis Results.
2. From the **View** menu, select **Load Animation**.
3. Double-click **rprof_fourpost**.
4. Use the **Dynamic Rotate** tool  to view the vehicle from a front corner.
5. Select the **Play** tool .

   The vehicle vibrates in response to the profile inputs. Depending on the seed for the pseudo-random number generator used in the Road-Profile Generation tool (by default the seed is based on the computer's clock), you may see the vehicle drift up or down, as if driving over a slight hill.
6. From the **View** menu, select **Load Plot**.
7. From the **File** menu, point to **Import**, and then select **RPC File**.
8. Right-click the **File to Read** text box and search for the file you created in Creating a Road-Profile Data File **tut_road.rsp**
9. Select **OK**.
10. From the **Channel** list, select both **LElev** and **RElev**.
11. Select **Add Curves**.

    Adams PostProcessor displays a plot similar to the one shown next.



With the parameters Ge=0.1, Gs=20, and Ga =0.1, you should see the LElev and RElev channels appear to be very similar on a large distance scale. If, however, you zoom in, you will see that they are very different. This is because, as with a real road, the low-frequency content of the wheeltracks is well correlated, but the high-frequency content is not. Note that your road might look different because it was probably generated with a different random-number seed than the one shown above.

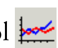
12. Return to Adams Car Ride.

Simulating a Single Bump

In this section, you set up the four-post test rig to accept inputs stored in tabular functions that you can create and modify using the Adams Car Curve Manager. You will use such a function to simulate the left side of the car driving over a bump in the road.

To set up the test rig:

1. After you've returned to Adams Car Ride, ensure that the **Full-Vehicle Analysis: Four Poster Testrig** is still open. If it is not, open it as you did in Setting up the Analysis.

2. Enter the following specifications:
    - **Output Prefix:bump**
    - **End Time:0.5**
    - **Target Value for Basis:500**
3. Select **Set Up Road Profiles**.
4. Set **Profile Source** to **table functions**.
5. Set **Vehicle Speed** to **100 km/h**.
6. For the **Left-Wheeltrack Profile**, right-click the **File Name** text box, and then search the **<aride_shared>** database for the file **bump_1inch.rpt**.
7. For the **Right-Wheeltrack Profile**, right-click the **File Name** text box, and then search the **<aride_shared>** database for the file **flat.rpt**.
8. Select the **Curve Manager** tool  to the right of the left-wheeltrack file name.

    The Curve Manager displays a plot with a 25.4 mm rectangular bump from x = 1 (m) to x = 2 (m), where x is the road station (the distance traveled down the road, projected into the global x-y plane).
9. To close the **Curve Manager**, select **Cancel**. You can ignore any warnings Adams Car Ride might issue.
10. In the **Road Profile Setup** dialog box, select **OK**.
11. In the **Full-Vehicle Analysis** dialog box, select **OK**.
12. Start Adams PostProcessor.
13. View the animation of **bump_fourpost** from the left-front corner of the vehicle. Note that only the left side of the vehicle experiences the bump.
14. Return to Adams Car Ride, and then exit the program.

Component Model Tutorial

Overview

This tutorial guides you through the process of running analyses on Adams Car Ride component models in isolation from other systems.

This chapter includes the following sections:

- Simulating a Hydromount with the Component Model Test Rig
- Deriving Hydromount-Model Parameters with the Isolator-Parameter Identification Tool

This tutorial takes about one hour to complete.

Simulating a Hydromount with the Component Model Test Rig

In this part of the tutorial you will load the Adams Car Ride plugin into Adams Car and learn how to perform a simulation using the Adams Car Ride Component-Model Test Rig. This test rig is for models of frequency-dependent force/moment producing elements, such as the Adams Car Ride Hydromount, the Adams Car Ride Frequency-Dependent Bushing, and the Adams Car Ride GSE Damper. The test rig is used to exercise the model in the same way that real physical elements are tested in the laboratory. You will use the test rig to examine the properties of a hydromount.

Opening the Assembly

You first open the assembly on which you will perform the analysis.

To open the assembly:

1. Start Adams Car Ride as explained in Starting Adams Car Ride.
2. From the **Ride** menu, point to **Component Analysis**, and then select **Component-Model Test Rig**.
3. Right-click the  tool and then select the  tool.
4. Right-click the **Assembly Name** text box, point to **Search**, and then select **<aride_shared>/assemblies.tbl**.
5. Double-click **component_hydro_bushing_example.asy**.
6. Select **OK**.
7. Close the Message Window.

Setting Up the Test Rig and Running Simulations

You will set up the test rig to exercise the hydromount at seven discrete frequencies. For each of the seven frequencies (Hz) 1, 2, 5, 10, 20, 50, and 100, you will specify that a simulation will be run with a prescribed sinusoidal Z displacement of the hydromount for each of the of the three amplitudes (mm) 1, 2, and 5. In addition, each sinusoidal input will have a phase angle of zero. Adams Car Ride will perform 21 (7x3) simulations and store all data in memory.

When all simulations are completed, Adams Car Ride will create requests from which you can create plots to show both the dynamic stiffness and loss angle of the hydromount as a function of frequency. You can create three such plots for dynamic stiffness, and three for loss angle. They will show dynamic stiffness versus frequency, and loss angle versus frequency, for each of the three amplitudes.

Each of the 21 simulations proceeds in time for up to ten cycles of the sinusoid. A simulation terminates earlier than this if steady-state behavior of the hydromount is achieved before the end of the tenth cycle via the energy sensor. 256 equally spaced data points are stored for each cycle of a sinusoid.

To set up the test rig and run simulations:

1. For getting the transfer functions calculated in the simulation results of the testrig, change the output settings for the solver: click on **Settings** - **Solver** - **Output** - **Request File** - **Yes**
2. From the Component Analysis dialog box, select **Set Up Test Rig** (located at the bottom).

   Adams Car Ride displays the Component Analysis: Set Up Test Rig dialog box. Note that Component Assembly is set to the same assembly you chose from the Component Analysis dialog box: `component_hydro_bushing_example.asy`.
3. Enter the following specifications:
   - **Actuation Type**: **Motion Driven**
   - In the area that follows **Motion Degrees of Freedom**, select the following:

	Constraint:	Initial:	Value:
X	Locked	Displacement	0.0
Y	Locked	Displacement	0.0
Z	Motion	Displacement	0.0
AX	Locked	Displacement	0.0
AY	Locked	Displacement	0.0
AZ	Locked	Displacement	0.0

   - The test rig is now set up so that it prescribes a displacement for the Z translation of the hydromount as a function of time. All other motions are locked, so only Z translation occurs. The initial values of all displacements are zero, meaning that the I and J markers that define the bodies connect to the hydromount.
4. Select **OK** to return to the **Component Analysis** dialog box.

5. Enter the following specifications:
   - **Output Prefix: hydro_test**
   - **Excitation Function:Set of Frequencies**
   - **Frequency:1, 2, 5, 10, 20, 50, 100**
   - **Maximal Cycles:10**
   - **Steps per Cycle:256**
   - **Axis: Z**
   - **Excitation Amplitudes 1, 2, 5**
   - **Phase:0.0**
   - **Loop Over: Frequency**
   - **Energy Sensor: On**
   - If Energy Sensor were set to Off, all 10 cycles would always be completed for each sinusoid.
   - **Measuring Method: Min-Max**

     Do not select **Keep Files.** If you select Keep Files, Adams Car Ride does not erase the data files for each of the 21 simulations.
6. Select **OK**.

   Running the simulations will take a few minutes.

Viewing the Analysis Results

You will now view plots of the dynamic stiffness and loss angle of the hydromount as a function of frequency.

To view the analysis results:

1. From the **Review** menu, select **Postprocessing Window**, or press **F8**.
2. In the dashboard, set **Source** to **Result Sets**.

   In the Simulation list you should see simulation results named:

   `.component_hydro_bushing_example.`*`{result}, where {result} is:`*

   `hydro_test_fre_sweep_1_1` through `hydro_test_fre_sweep_7_1`, `hydro_test_fre_sweep_1_2` through `hydro_test_fre_sweep_7_2`, and `hydro_test_fre_sweep_1_3` through `hydro_test_fre_sweep_7_3`.

   The first index in the simulation name is the i^{th} frequency of the sweep. Therefore, i = 1, 2, 3, 4, 5, 6, and 7 correspond to the frequencies (Hz) 1, 2, 5, 10, 20, 50, and 100, respectively. The second index in the simulation name is the j^{th} amplitude of the sweep. Therefore, j = 1, 2, and 3 correspond to amplitudes (mm) 1, 2, and 5, respectively. For each group of simulations of the same amplitude (j = 1, 2, or 3), there is also a result set named hydro_test_Transfer_Function_j. Therefore, you should see:

   `hydro_test_Transfer_Function_1`

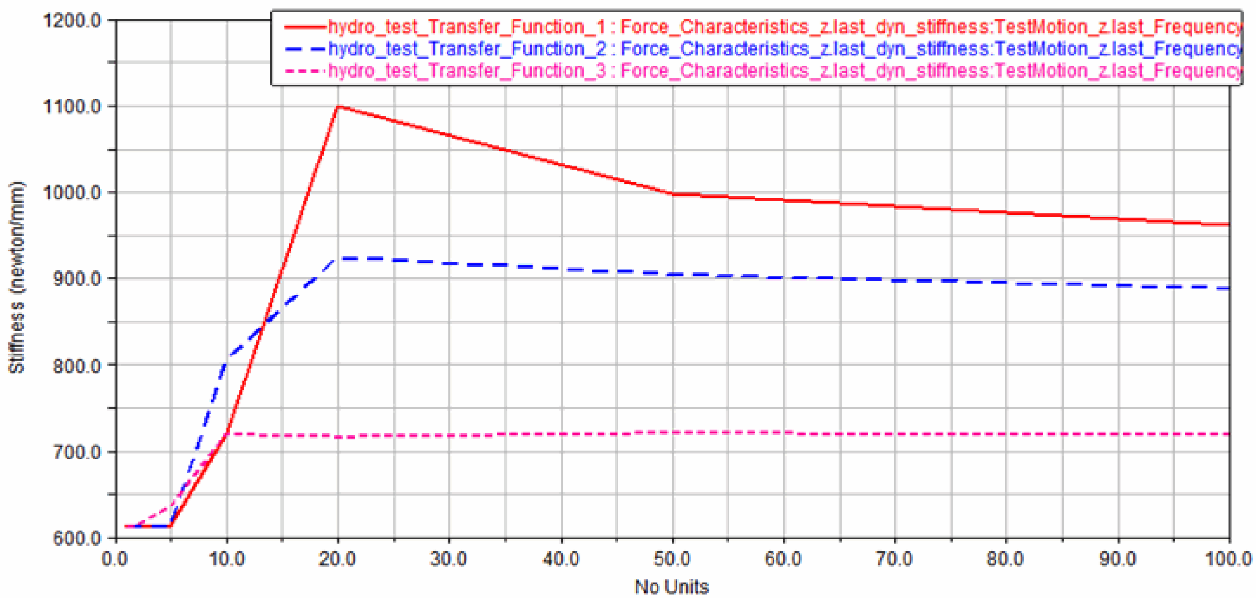
```
hydro_test_Transfer_Function_2
hydro_test_Transfer_Function_3
```

The transfer_function results sets contain the steady-state data for all frequencies at a given amplitude.

Note:	Solver output settings for the request files has to be switch on for getting the transfer functions (**Settings - Solver - Output - Request File - Yes**)

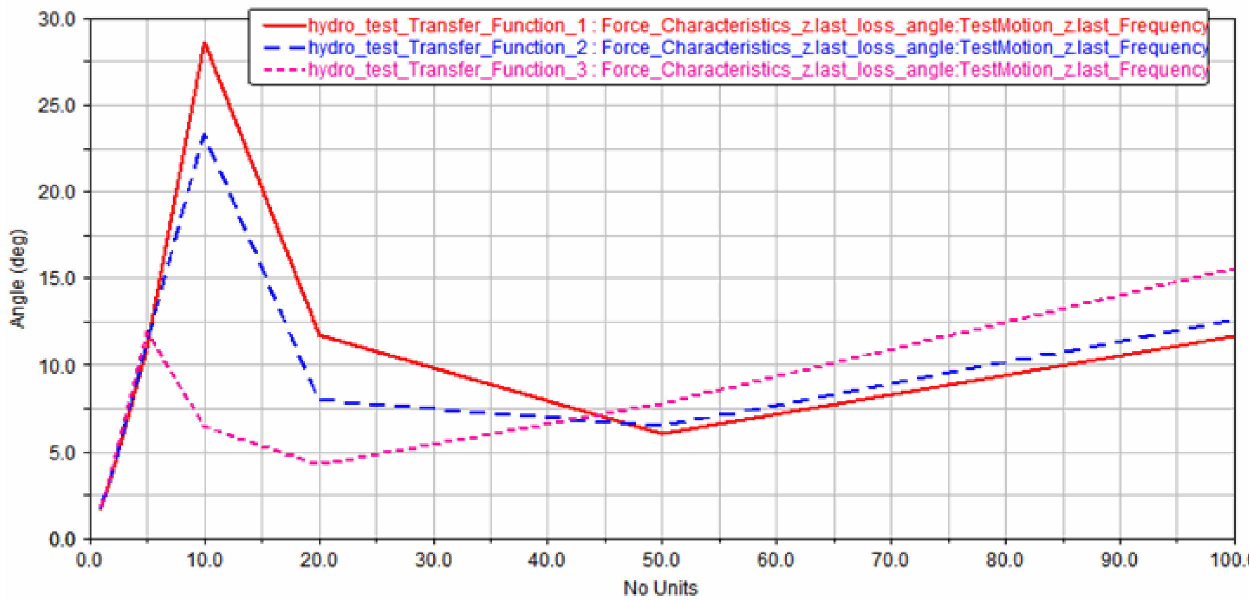
3. From the **Simulation** list, select **hydro_test_Transfer_Function_1**.
4. From the **Result Set** list, select **Force_Characteristics_z**.
5. Set **Independent Axis** to **Data** (on the bottom right side of the dashboard).
6. In the Independent Axis Browser:
   - From the **Result Set** list, select **TestMotion_z.**
   - From the **Component**, select **last_freq**.
   - Select **OK**.
7. In the dashboard, from the **Component** list, select **last_dyn_stiffness**.
8. Select **Add Curves**. You now have a plot of dynamic stiffness versus frequency for a sinusoidal excitation amplitude of 1 (mm).
9. Repeat steps 3 through 8 for the entries **hydro_test_Transfer_Function_2** and **hydro_test_Transfer_Function_3**, adding the curves to the same plot. This yields a plot with three curves of dynamic stiffness versus frequency, each for a different excitation amplitude. Your plot should look similar to the one shown next.

Note:	You can drag the legend to move it away from the plot peak.



10. Select the **New Page** tool.

11. Now repeat steps 3 through 8 for the entries **hydro_test_Transfer_Function_1**, **hydro_test_Transfer_Function_2**, and **hydro_test_Transfer_Function_3**, again adding the curves to the same plot. This time, however, select **last_loss_angle** when choosing the data for the vertical axis from the **Component** list in the dashboard. This yields a plot with three curves of loss angle versus frequency, each for a different excitation amplitude. Your plot should look similar to the one shown next.



12. To return to Adams Car Ride, press **F8**.

    Now that you've completed this part of the tutorial, you can run similar tests on the GSE damper model and other components.

Deriving Hydromount-Model Parameters with the Isolator-Parameter Identification Tool

In the first part of this tutorial, you used the Adams Car Ride Component-Model Test Rig to calculate the dynamic stiffness and loss-angle characteristics of a hydromount for a range of sinusoidal excitation frequencies and amplitudes that you specified. The Adams Car Ride Hydromount Component Model uses eight parameters to model a hydromount. These eight parameters for a hydromount produce the response you viewed in the first part of the tutorial. These parameters, however, must be obtained before the model can be used.

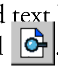
To obtain the parameters, you use the Adams Car Ride Isolator-Parameter Identification Tool (IPIT). Given a set of hydromount test data, the IPIT finds values for the eight model parameters, such that the model reproduces the test data within a specified error tolerance. In this part of the tutorial, you will use the IPIT to do that.

Opening the Hydromount Property File

To run IPIT, you must launch IPIT and open the file containing the hydromount test data. The IPIT uses this as input for the parameter derivation. Before running IPIT, however, you should know something about the structure of a hydromount property file. So, first you'll open an example property file and take a look at the contents.

To view the parameter file:

1. If Adams Car Ride is not running, start it as explained in Starting Adams Car Ride.
2. If the **component_hydro_bushing_example.asy** is still open, skip to step 3. Otherwise, open the **component_hydro_bushing_example.asy** as you did in Opening the Assembly.
3. If the **component_hydro_bushing_example.asy** is not in view:
   - From the **View** menu, select **Assembly**.
   - From the Assembly pull-down menu, select **component_hydro_bushing_example.asy**.
   - Select **OK**.
4. Right-click the hydromount, which is the dark gray cylinder in the center of the screen, point to **Hydro_bushing: component_hydro_bushing.bgs_b1**, and then select **Modify**.

   Adams Car Ride displays the Modify Hydro Bushing dialog box. Note that there are two Property File labels in the dialog box:
   - The text box adjacent to the first label displays the path to the file currently being used for the hydromount: `mdids://aride_shared/hydro_bushing.tbl/hyd_bus001.hbu`.
   - The second one doesn't have an associated text box, but displays an **Apply Property File** button to the right of the **View Property File** tool .
5. Select the **View Property File** tool to the right of the second **Property File** label.

   The Information window appears, displaying the contents of the property file. Look through the file and note the six blocks of data: `[MDI_HEADER]`, `[UNITS]`, `[GENERAL]`, `[HYDRO_PARAMETERS]`, `[HYDRO_IDENTIFICATION_DATA]`, and `[HYDRO_TEST_DATA]`. We will focus on the last three blocks:
   - The `[HYDRO_PARAMETERS]` block contains the eight parameters used for the hydromount model. These were used when you performed the simulations in the first part of this tutorial.
   - The data in the `[HYDRO_TEST_DATA]` block is meant to be obtained directly from testing a real hydromount in the laboratory. It is not used directly by the hydromount model. It is, however, used by the IPIT.
   - The data in the `[HYDRO_IDENTIFICATION_DATA]` block is calculated using the hydromount model with the parameters in the `[HYDRO_PARAMETERS]` block. There is exactly one data point for each data point in the `[HYDRO_TEST_DATA]` block. Ideally, the `[HYDRO_IDENTIFICATION_DATA]` block and the `[HYDRO_TEST_DATA]` would be identical. This would be the case if the hydromount model were perfect and the parameters in the `[HYDRO_PARAMETERS]` block were chosen perfectly. It is the job of IPIT to choose the parameters so that the identification data matches the test data (within user-specified tolerances).

At a minimum, IPIT requires only the `[HYDRO_TEST_DATA]` block to run (along with the `[MDI_HEADER]`, `[UNITS]`, and `[GENERAL]` blocks). In this case, it starts with all parameters set to zero. An optimizer within IPIT tries to select parameters so that identification data matches test data within user-specified tolerances. When the optimizer is done, the IPIT saves a new property file. This file contains parameters it derived and identification data implied by the model given those parameters.

You can then use this new file as input to the IPIT on a subsequent run. In this case, IPIT starts the optimization with the parameters in the `[HYDRO_PARAMETERS]` block, so it can begin where it left off. This is useful if you want to derive new parameters that match test data with tighter tolerances.

Next you will make a new property file without any parameters to use as input to IPIT, and then you will run IPIT with this file.

6. Close the Modify Hydro Bushing dialog box.

To make a new property file and run the IPIT:

1. Using a text editor, open the file **mdids://aride_shared/hydro_bushing.tbl/hyd_bus001.hbu**. Save a copy called **mdids://aride_shared/hydro_bushing.tbl/hyd_bus001_test_data_only.hbu**. If not already opened, open **mdids://aride_shared/hydro_bushing.tbl/hyd_bus001_test_data_only.hbu** in a text editor.
2. Delete the blocks [**HYDRO_PARAMETERS**] and [**HYDRO_IDENTIFICATION_DATA**] and all associated data, including the $---... line that starts each block.
3. Change the name of **BUSHING_PROPERTY_FILE** to **mdids://aride_shared/hydr_bushings.tbl/mdi_0001_test_data_only.bus**.
4. Save **mdids://aride_shared/hydro_bushing.tbl/hyd_bus001_test_data_only.hbu**, and then exit the text editor.

   Now you are ready to run IPIT.
5. Note that you can launch the IPIT tool from the interface. From the **Ride** menu, point to **Tools**, and then select **Isolator-Parameter Identification**).
6. Select **Load**, and then browse for the file you just created: **mdids://aride_shared/hydro_bushing.tbl/hyd_bus001_test_data_only.hbu**.

   After the file loads, you will see that all parameters are set to zero. The Plot tab should be active. In it you will see two scatter plots of the test data: one for dynamic stiffness (the top plot) and one for loss angle (the bottom plot).
7. Select the **Data** tab. You should see the contents of the file you just created.
8. Select the **Plot** tab to display the plots again. Leave the defaults values for the **Error Control** parameters unchanged.
9. Select **Start Optimization**.

   The optimizer begins running. As it progresses, you will see solid-line plots of the identification data, generated with the current values of the hydromount parameters, superimposed upon the test data.

   With the default error control parameters and the sample test data, it takes quite a while for the optimization to finish.

The optimizer will stop after completing the optimizer has found the optimal solution. At the bottom of the dialog box, you will see the following message: Hydromount HBU optimization (1) finished, objective function value:

10. When the message at the bottom of the dialog box changes to **Ready to Start Optimization**, or **Calculate Frequency Response data**, select **Save**.

    By default, IPIT makes a file name for output that is the same as the input file name, but with `_out` appended to the root name.
11. Select **Load** and load the output file you just saved: **mdids://aride_shared/hydro_bushing.tbl/hyd_bus001_test_data_only_out.hbu**.

    You will see the same parameters that were just calculated by the optimizer.
12. Select the **Data** tab.

    You will see that the file created as output contains the hydromount parameters and the identification data.
13. Select **Quit**.
14. When asked to save your data, select **No**.
15. Exit the program.

Note:	One can run a second Hydromount optimization step (2: Hydromount (time domain)), which is running an optimization using simulations of the component testrig. This optimization takes longer, but may decrease the differences in between the hydromount model and the testdata.