MotionDesk

Scene Animation

For MotionDesk 4.8

Release 2021-A – May 2021

dSPACE

How to Contact dSPACE

Mail:	dSPACE GmbH Rathenaustraße 26 33102 Paderborn Germany
Tel.:	+49 5251 1638-0
Fax:	+49 5251 16198-0
E-mail:	info@dspace.de
Web:	http://www.dspace.com

How to Contact dSPACE Support

If you encounter a problem when using dSPACE products, contact your local dSPACE representative:

- Local dSPACE companies and distributors: http://www.dspace.com/go/locations
- For countries not listed, contact dSPACE GmbH in Paderborn, Germany.
  Tel.: +49 5251 1638-941 or e-mail: support@dspace.de

You can also use the support request form: http://www.dspace.com/go/supportrequest. If you are logged on to mydSPACE, you are automatically identified and do not need to add your contact details manually.

If possible, always provide the relevant dSPACE License ID or the serial number of the CmContainer in your support request.

Software Updates and Patches

dSPACE strongly recommends that you download and install the most recent patches for your current dSPACE installation. Visit http://www.dspace.com/go/patches for software updates and patches.

Important Notice

Contents

Reference Information 135

About This Document

Contents

This document introduces you to the animation in MotionDesk. It provides all the information required for animating the scene in a simulation.

Symbols

dSPACE user documentation uses the following symbols:

Symbol	Description
DANGER	Indicates a hazardous situation that, if not avoided, will result in death or serious injury.
WARNING	Indicates a hazardous situation that, if not avoided, could result in death or serious injury.
CAUTION	Indicates a hazardous situation that, if not avoided, could result in minor or moderate injury.
NOTICE	Indicates a hazard that, if not avoided, could result in property damage.
Note	Indicates important information that you should take into account to avoid malfunctions.
Tip	Indicates tips that can make your work easier.
	Indicates a link that refers to a definition in the glossary, which you can find at the end of the document unless stated otherwise.
	Precedes the document title in a link that refers to another document.

Naming conventions

dSPACE user documentation uses the following naming conventions:

%name% Names enclosed in percent signs refer to environment variables for file and path names.

< > Angle brackets contain wildcard characters or placeholders for variable file and path names, etc.

Special folders

Common Program Data folder A standard folder for application-specific configuration data that is used by all users.

`%PROGRAMDATA%\dSPACE\<InstallationGUID>\<ProductName>`

or

`%PROGRAMDATA%\dSPACE\<ProductName>\<VersionNumber>`

Documents folder A standard folder for user-specific documents.

`%USERPROFILE%\Documents\dSPACE\<ProductName>\<VersionNumber>`

Local Program Data folder A standard folder for application-specific configuration data that is used by the current, non-roaming user.

`%USERPROFILE%\AppData\Local\dSPACE\<InstallationGUID>\<ProductName>`

Accessing dSPACE Help and PDF Files

After you install and decrypt dSPACE software, the documentation for the installed products is available in dSPACE Help and as PDF files.

dSPACE Help (local) You can open your local installation of dSPACE Help:

- On its home page via Windows Start Menu
- On specific content using context-sensitive help via **F1**

dSPACE Help (Web) You can access the Web version of dSPACE Help at www.dspace.com/go/help.

To access the Web version, you must have a *mydSPACE* account.

PDF files You can access PDF files via the  icon in dSPACE Help. The PDF opens on the first page.

Basics and Instructions

Where to go from here

Information in this section

Data Source for Motion Data

Introduction

To animate the scene, MotionDesk must be supplied with motion data.

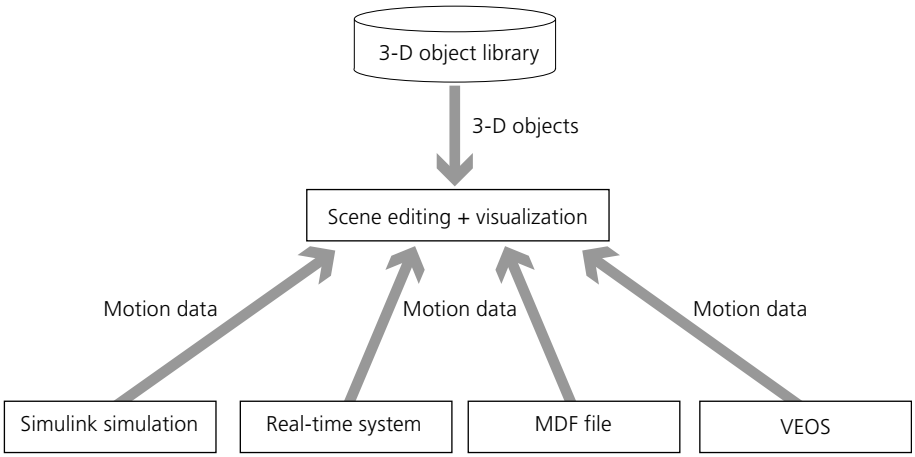
Where to go from here

Information in this section

Basics on Selecting the Data Source

Introduction

To animate the scene, MotionDesk must be supplied with motion data, see the following illustration.



MotionDesk can get the motion data from several types of data sources, for example, a real-time application, a PC-based Simulink simulation, VEOS, or a motion data file (MDF file). One of these data sources must be selected in MotionDesk.

Real-time application

A real-time application runs on a real-time system and simulates the movement of a mechanical system, for example, a car driving on a test track. The real-time application must include the MotionDesk services, which calculate the motion

data from the simulation values and write them to a buffer. The buffer is read by MotionDesk, which animates the movable objects correspondingly.

Simulink simulation and VEOS

The process is the same as described above for the real-time application, with the exception that there is no real-time hardware. Instead, a PC-based Simulink simulation or VEOS runs on a host PC.

MDF file

The motion data coming from a real-time application or model can be stored in an MDF file. MotionDesk can read the file and replay the animation without the need of a real-time system or simulator.

How to Select the Data Source

Objective

To get motion data for the movable objects, you must select the data source where the motion data is calculated.

Basics

Multi-PC solution For basic information on network settings and instructions to install a multi-PC solution, refer to Setting Up a Multi-PC Solution (MotionDesk Calculating and Streaming Motion Data 🕮).

Simulink simulation For instructions on installing a system to visualize a PC-based Simulink simulation, refer to Setting Up Visualization for a Simulink Simulation (MotionDesk Calculating and Streaming Motion Data 🕮).

Automatic animation start If the simulation system used as the data source is connected to ControlDesk or AutomationDesk, the animation can be started automatically when the real-time simulation starts.

Data sources

MotionDesk can receive the motion data from several types of simulation systems. You have to select the data source according to your simulation system. There are different methods to select the data source depending on the simulation system.

You can select the motion data source in the **Motion Player**.

Tip

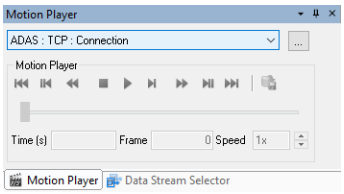
You can also select the source in the list in **Platform - Sources** on the **Home** ribbon.
You select **Customize** to configure the platform or motion data source.
For more information, refer to: Customize Data Source on page 172.

Simulation System	Data Source	Refer to ...
ADAS	ADAS : TCP : Connection	Method 1
Modular hardware based on a DS1007	DS1007 : UDP : Connection	Method 2
MDF file	File : MDF : <Stream>	Method 3
Real-time board connected via bus connection (DS1006)	Hardware : Type : <Stream>	Method 4
Real-time board connected via network connection (MicroAutoBox II or modular hardware with slot CPU)	Hardware-Net : Type : <Stream>	Method 5
MicroAutoBox III	MABX III : UDP : Connection	Method 6
MicroLabBox	MLBX : UDP : Connection	Method 6
Multi-PC solution (DS1006-based systems only)	Network : UDP : Connection	Method 7
Simulink simulation (1 or 2 PCs)	Network : UDP : Connection	Method 7
SCALEXIO system (SCALEXIO Processing Unit or DS6001 Processor Board)	SCALEXIO : UDP : Connection	Method 8
VEOS	VEOS : UDP : Connection	Method 9

Method 1

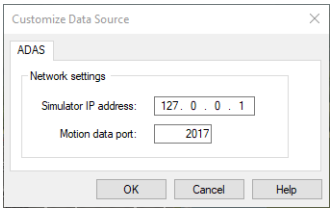
To select an ADAS simulator as the data source

1 In the Motion Player, select ADAS : TCP : Connection or MABX III : UDP : Connection as the data source.



2 Click the Browse button.

MotionDesk opens the Customize Data Source dialog for selecting an ADAS simulator, see the following illustration.



3 Specify the IP address of the ADAS simulator on the network.

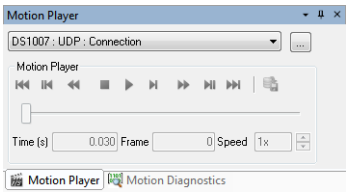
4 Specify the motion data port number of the data from the selected ADAS simulator on the network.

5 Click OK to confirm your selection.

Method 2

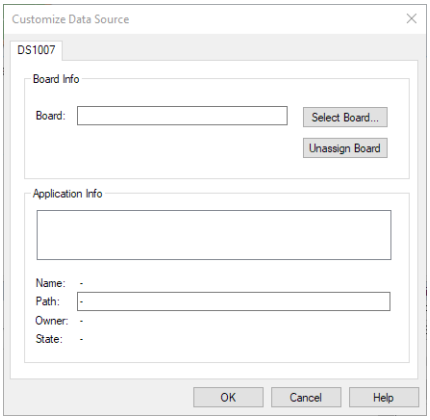
To select a DS1007 as the data source

1 In the Motion Player, select DS1007 : UDP : Connection as the data source.



2 Click the Browse button.

MotionDesk opens the Customize Data Source dialog for selecting a DS1007 board, see the following illustration.



3 Click Select Board.

MotionDesk opens a dialog which lists all DS1007 boards that are connected to the network.

4 Select a DS1007 board and click OK to close the dialog.

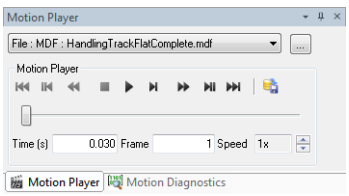
The Application Info field gives you information on the real-time application of the DS1007 board.

5 Click OK to confirm your selection.

Method 3

To select an MDF file as the data source

1 In the Motion Player, select File : MDF : <Stream> as the data source.



2 Click the Browse button.

MotionDesk opens a standard open dialog.

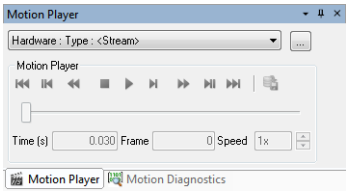
3 In the Open dialog, select an MDF file and click Open.

MotionDesk reads the MDF file and writes the motion data to the frame buffer.

Method 4

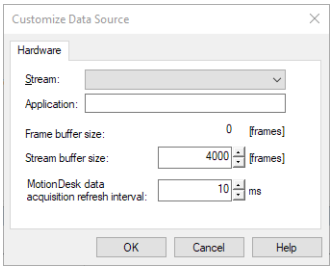
To select the data source for a 1-PC solution

1 In the Motion Player, select Hardware : Type : <Stream> as the data source.



2 Click the Browse button.

MotionDesk opens the Customize Data Source dialog for a real-time board, see the following illustration.



3 In the Stream list, select the processor board on which the MotionDesk real-time service is running.

The name of the running application is displayed in the Application field.

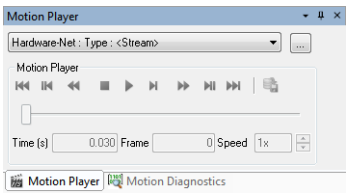
4 In the Stream buffer size edit field, enter the number of frames to be buffered.

5 Click OK to confirm your selection.

Method 5

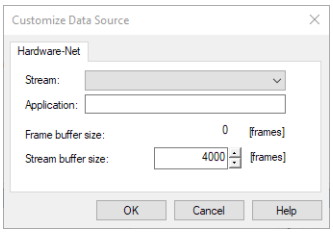
To select the data source for a network connection

1 In the Motion Player, select Hardware-Net : Type : <Stream> as the data source.



2 Click the Browse button.

MotionDesk opens the Customize Data Source dialog for a real-time board, see the following illustration.



3 In the Stream list, select the real-time board on which the MotionDesk real-time service is running.

The name of the running application is displayed in the Application field.

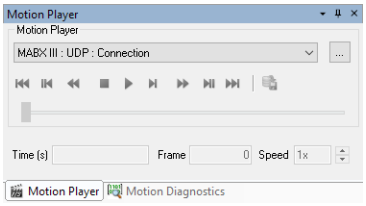
4 In the Stream buffer size edit field, enter the number of frames to be buffered.

5 Click OK to confirm your selection.

Method 6

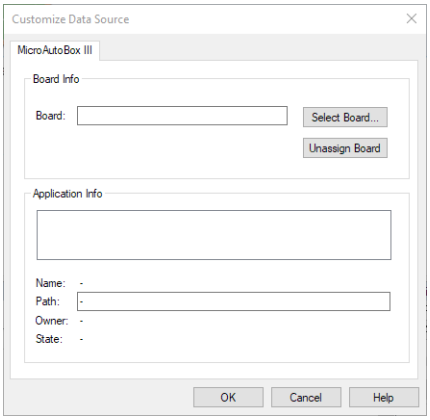
To select a MicroAutoBox III or MicroLabBox as the data source

1 In the Motion Player, select MABX III : UDP : Connection or MLBX : UDP : Connection as the data source.



2 Click the Browse button.

MotionDesk opens the Customize Data Source dialog for selecting a MicroAutoBox III or MicroLabBox, see the following illustration.



3 Click Select Board.

MotionDesk opens a dialog which lists all MicroAutoBox III or MicroLabBoxes that are connected to the network.

4 Select the MicroAutoBox III or MicroLabBox and click OK to close the dialog.

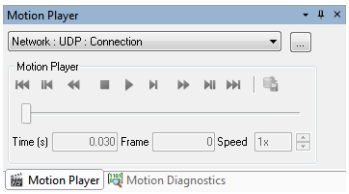
The Application Info field gives you information on the real-time application of the MicroAutoBox III or MicroLabBox.

5 Click OK to confirm your selection.

Method 7

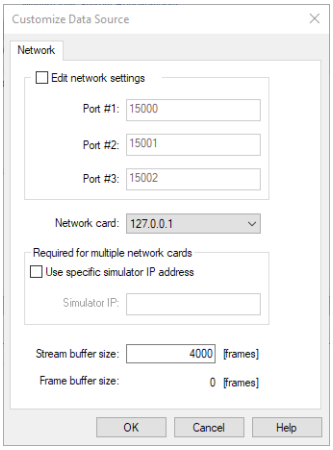
To select the data source for a Simulink simulation or a multi-PC solution

1 In the Motion Player, select Network : UDP : Connection as the data source.



2 Click the Browse button.

MotionDesk opens the Customize Data Source dialog for the network settings, see the following illustration.



3 Select Edit Network Settings.

4 Set Port #1 to #3 to the same values as defined in the MD_Communication block in your simulation model.

5 In the Network Card list, select the IP address. If you have more than one network adapter in your MotionDesk PC, make sure that you select the one used for your local visualization network.

6 In the Stream buffer size edit field, enter the number of frames to be buffered.

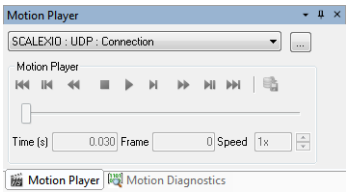
7 In the Frame buffer size, define the possible number of frame buffers.

8 Click OK to confirm the settings.

Method 8

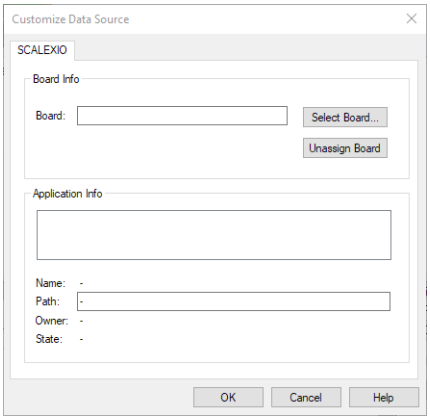
To select a SCALEXIO system as the data source

1 In the Motion Player, select SCALEXIO : UDP : Connection as the data source.



2 Click the Browse button.

MotionDesk opens the Customize Data Source dialog for selecting a SCALEXIO system, see the following illustration.



3 Click Select Board.

MotionDesk opens a dialog which lists all SCALEXIO system which are connected to the network.

4 Select the SCALEXIO system and click OK to close the dialog.

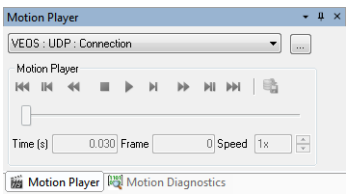
The Application Info field gives you information on the real-time application of the SCALEXIO system.

5 Click OK to confirm your selection.

Method 9

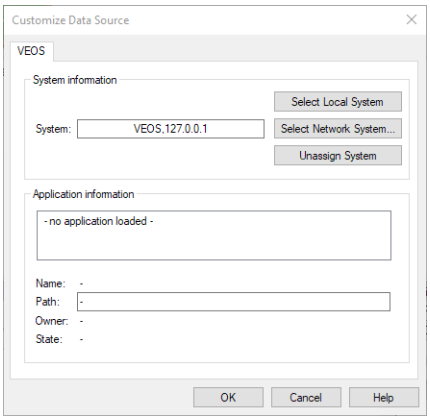
To select VEOS as the data source

1 In the Motion Player, select VEOS : UDP : Connection as the data source.



2 Click the Browse button.

MotionDesk opens the **Customize Data Source** dialog for selecting a VEOS, see the following illustration.



3 To select a VEOS running on the same PC as MotionDesk, click **Select Local System**.

If a VEOS is found, MotionDesk displays information on the VEOS in **System** and information on the application in **Application Info**.

4 To select a VEOS that is connected via network to the MotionDesk,

1. Click **Select Network System**.

   MotionDesk opens a dialog which lists all VEOS found.
2. Select a VEOS and click **OK** to close the dialog.

MotionDesk displays information on the VEOS in **System** and information on the application in **Application Info**.

5 Click **OK** to confirm your selection.

Result

The data source is selected.

Next step

You can set up the movable objects, refer to Setting up Movable Objects on page 24.

Related topics

Basics

Data Source for Motion Data 15

References

Customize Data Source 172
Motion Player 184

Setting up Movable Objects

Introduction

When the data source for motion data has been selected, the movable objects can be set up.

Where to go from here

Information in this section

Basics of Setting up Movable Objects

Introduction

Movable objects are the 3-D objects which positions and orientations are calculated in a simulation.

Setting up movable objects

When the data source for motion data has been selected, the movable objects can be set up in the following order:

1. Insert 3-D objects into the scene that are used as movable objects. Refer to How to Insert Movable Objects Into the Scene on page 25.
2. Assign motion data to every movable object. Refer to How to Assign Motion Data to Movable Objects on page 26.
3. Start the simulation or replay of a motion data file to get a valid initial position. Refer to How to Start the Animation on page 51.
4. Adjust the relative coordinate systems of the movable objects to move them to correct positions or orientations. Refer to How to Adjust Movable Objects on page 28

How to Insert Movable Objects Into the Scene

Objective

Inserting movable objects into the scene is simple.

Basics

Movable objects are the 3-D objects that move in the scene. The position and orientation of movable objects are calculated in a simulation. It is therefore not necessary to specify these settings manually. You can adjust them by specifying an offset value for the calculated values.

When 3-D objects are in the 3-D Object Library, they have the Create as property. This property indicates whether a 3-D object is a movable or static object by default. However, it is also possible to insert a static object as a movable object in the scene and vice versa.

The dSPACE 3-D object library includes a range of road, scenery, and vehicle objects that can be added to a MotionDesk scene. The vehicles include specific models of Mercedes, BMW, Volkswagen, and NCAP global vehicle targets (GVT). Lorries, trailers and roadside assistance and emergency service vehicles are also included. You can assign a motion data stream from the simulation to the movable objects.

Precondition

A project must have been created and an experiment must be active.

Method

To insert movable objects into the scene

1. In the Library Browser, choose a 3-D object.
2. Depending on the type of the 3-D object, do the following:
   - If the object is a movable object by default, drag it into the 3-D View with the left mouse button.

     The movable object is inserted.
   - If the object is a static object by default, drag it into the 3-D View with the right mouse button.

     In the dialog that opens, select Create as movable object.

     The object is inserted as a movable object.

Result

You have inserted movable objects.

Related topics

Basics

3-D Object Libraries (MotionDesk Scene Creation 🕮)

How to Assign Motion Data to Movable Objects

Objective

The movable objects in the scene must be supplied with motion data from a simulation or a motion data file. For more information, refer to Data Source for Motion Data on page 15.

Possible methods

If you select a new data source, the motion data must be assigned to the movable objects. There are two ways to do this:

- As the assigned motion data is one property of a movable object, you can select a data stream in the **Properties** pane. Refer to Method 1 on page 26.
- Another method to assign motion data is to use the **Data Stream Selector**. This way is faster than using the **Object Properties** dialog. Refer to Method 2 on page 26.

Method 1

To assign motion data to movable objects via the Properties pane

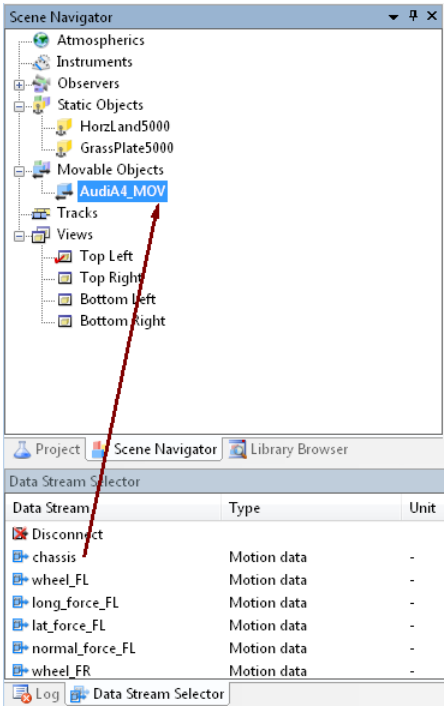
1. On the **Scene Navigator**, select a movable object.
   The **Properties** pane displays the properties of the selected movable objects.
2. In the **Motion data** property, select the data stream.
3. Repeat the above steps for all movable objects.

Method 2

To assign motion data to movable objects via drag & drop

1. In the **Data Stream Selector**, click a **data stream** and drag it to the appropriate movable object in the **Scene Navigator**.

If you want to disconnect an object from its motion data stream, drag **Disconnect** from the data stream list to the object in the **Scene Navigator**.



Tip

If you want to change the assigned motion data, just drag the new data stream to the movable object. It is not necessary to disconnect the existing connection first.

2 Repeat the above steps for all movable objects.

Result

As the movable objects were placed anywhere in the scene, their initial positions do not match the simulated values. You do not have to move the objects to valid positions manually. The objects will be positioned by the motion data of the data stream.

Related topics

Basics

Data Source for Motion Data..... 15

HowTos

How to Adjust Movable Objects..... 28
How to Select the Data Source..... 16

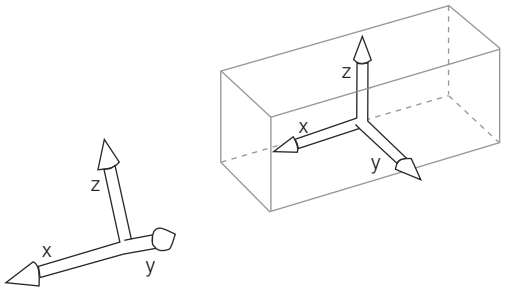
How to Adjust Movable Objects

Objective

Adjusting the movable objects means defining offset values for their positions and orientations which are added to the motion data.

Basics

The adjustment moves and rotates the origin of each visualized object to match the origin of the coordinate system used in the simulation, see the following illustration.

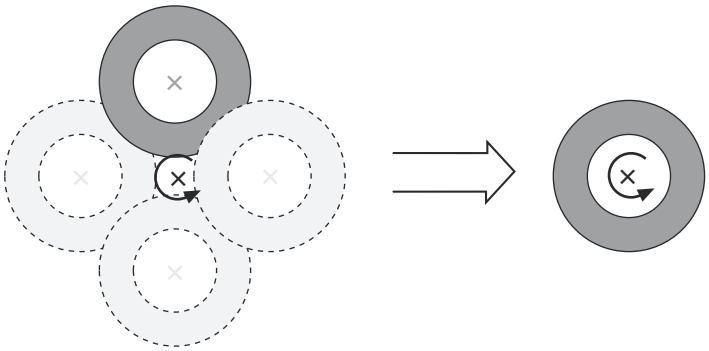


Adjusting a movable object can be necessary

- If the coordinate system of the simulation model differs from the CAD system that was used for designing the movable object, or
- If the dimensions of the simulated object and the movable object differ.

In MotionDesk the coordinate systems of movable objects can be moved and rotated to adjust to the coordinate system used in the simulation. MotionDesk moves each movable object according to the translation parameters (x, y, z) first and then rotates it according to the rotation parameters (roll, pitch, yaw).

Because the translation is done before the rotation, the animation of a rotating object (for example, a wheel) is eccentric when you change its translation. If an object is rotating eccentrically, you can use the translation parameters to get rid of the eccentricity, see the following illustration:



Ensure that your adjustments are suitable; in other words, do not let a car that has a short distance between its axles in the simulation have a long distance between them in the visualization. In that case the behavior of the movable object calculated from the simulation would not match the visualization. Instead

of this you can scale the x-axis of the car's chassis to get the chassis's wheel house in the position of the centric rotating wheels.

In the object's Properties pane, you can enter offset values for the position and orientation. The modifications are immediately visible in the 3-D View.

Method

To adjust movable objects

1 On the Scene Navigator, select a movable object.

The Properties pane displays the properties of the movable object. The selected object is displayed in a bounding box.

2 In the Properties pane, specify a value for the offset position and rotation.

3 You can scale the movable object if it is not the same size as the simulated object. Enter the scaling values in the Properties pane.

4 Movable objects can be displayed with different rendering techniques, which you can select in the Properties pane. If you select a more realistic rendering technique, the computation time increases.

The 3-D View displays the object with the selected rendering technique only if the scene is displayed in the mixed view.

Note

Transparent Objects

Transparent objects placed inside other transparent objects can disappear in Simple Lighting mode. Weather conditions, for example, rain, and snow can also be hidden behind transparent objects. This is due to a graphics limitation.

You can switch Advanced Lighting. For troubleshooting, refer to Transparent Objects and Weather Conditions are Hidden (MotionDesk Scene Creation 🕮).

5 Repeat the above steps for all the movable objects.

Result

The movable objects are adjusted.

Related topics

Basics

Cardan Roll, Pitch, and Yaw Angles (MotionDesk Calculating and Streaming Motion Data 🕮)
Coordinate System Used in MotionDesk (MotionDesk Calculating and Streaming Motion Data 🕮)

HowTos

How to Select the Data Source..... 16

References

Block Description (MD_Rotation_Angles) (MotionDesk Calculating and Streaming Motion Data 🕮)
Observer Properties..... 200

Using Instruments in the Scene

Introduction

You can use instruments to observe the values of simulation variables of the real-time application in the scene. For example, you can create a speedometer for a virtual car.

Where to go from here

Information in this section

Basics on Using Instruments in the Scene

Introduction

The purpose of the instruments is to display simulation variables such as the forces affecting a vehicle, the speed, or the selected gear.

Features of instruments

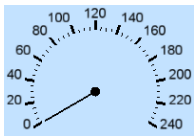
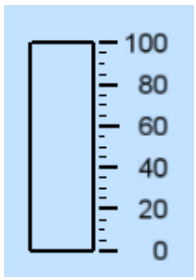
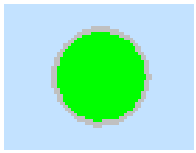
The instruments in MotionDesk provide the following features:

- Up to 300 variables can be visualized by instruments.
- Instrument data can be saved and loaded as MDF files.
- The number of instruments in MotionDesk is not limited.
- Six instrument types are available: Numeric, Gauge, Bar, Multistate LED, Multistate Picture, and Multistate Text.
- Several properties of the instruments can be specified, for example, size, position, color.

- The instrument's position can be specified in pixel or as a percentage value of the view.
- Instruments can be attached to movable objects or given a static position in the 3-D View.
- Instruments are linked to one or more observers.
- Several instruments can be grouped in an Instrument Panel.

Instrument types

Four instruments are available in MotionDesk, see the following table.

Instrument Type	Description
	The Numeric instrument is for numeric display and can be given a caption and a physical unit.
	The Gauge instrument consists of a scale and a needle to indicate the variable to be simulated. An optional numeric display can be added.
	The Bar instrument is a slider bar indicating the variable to be simulated. An optional numeric display can be added.
	The Multistate LED instrument shows a color evaluation of the variable to be simulated. It can visualize certain states, ranges, or thresholds of a variable with colored LEDs. An optional numeric display can be added.
	The Multistate Picture instrument shows different pictures in relation to the value of the variable to be simulated. It can visualize certain states, ranges, or thresholds of a variable displaying different pictures. An optional numeric display can be added.
	The Multistate Text instrument shows different strings in relation to the value of the variable to be simulated. It can visualize certain states, ranges, or thresholds of a variable displaying different strings. You can use macros to insert information on the variable, such as the value, into the string.

Workflow

To use instruments in the scene two steps are necessary:

1. Extending the real-time model

   The **MD_Instrumentation** block must be added to the real-time model in MATLAB/Simulink. All the variables to be visualized in an instrument must be connected to the block. Refer to How to Prepare the Real-Time Model for Using Instruments or State Objects (MotionDesk Calculating and Streaming Motion Data 🕮).

2. Creating instruments

   In MotionDesk, you create the instruments that display the values of the variables connected to the **MD_Instrumentation** block. There are several options, for example, the instruments can move together with a movable object or be static in the scene. For details on creating and specifying the instruments, refer to How to Use Instruments in the Scene on page 33.

Limitations when using instruments

The multi-track mode is not supported for instruments in MotionDesk.

MotionDesk's instruments do not support script fonts, such as the *Vladimir Script* font. You can identify script fonts by their name.

Related topics

Basics

Basics on Using Instrument Panels in the Scene.....36

How to Use Instruments in the Scene

Objective

You can use instruments in the scene to display the values of simulation variables.

Managing instruments

The instruments are managed in the **Scene Navigator**. The **Scene Navigator** has an **Instruments** folder that lists all the instruments of the scene. The menu commands for managing instruments are in the context menus of the **Instruments** folder and the instruments entries.

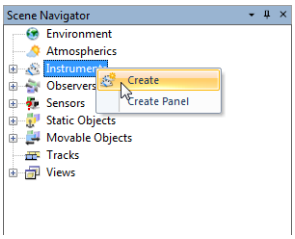
Preconditions

The real-time model must contain the **MD_Instrumentation** block and the simulation variables to be displayed must be connected to the block. Refer to How to Prepare the Real-Time Model for Using Instruments or State Objects (MotionDesk Calculating and Streaming Motion Data 🕮).

Method

To use instruments in the scene

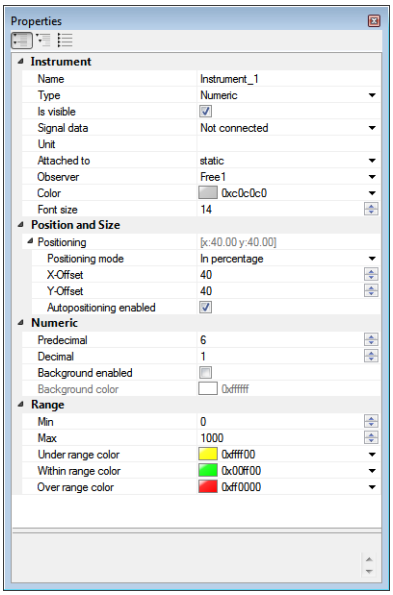
1 In the Scene Navigator, open the context menu of the Instruments folder and choose Create.



Or

On the Observation ribbon, click Create Instrument – <Instrument Type> (The selected instrument type is initially used, you can alter the type later).

MotionDesk creates a new instrument and displays its properties in the Properties pane.



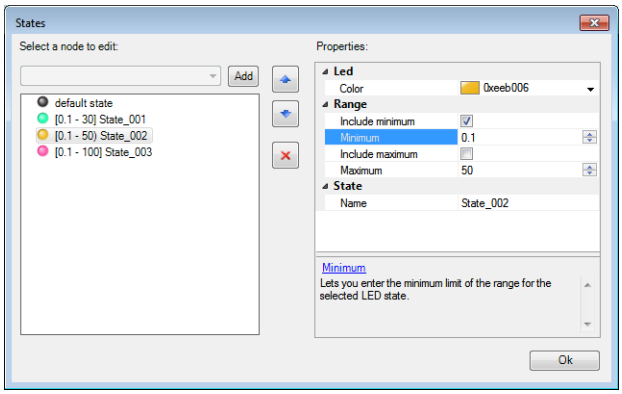
2 In the Properties pane, select the instrument type and specify its properties. For details on the properties, refer to Instrument Properties on page 192.

As long as the instrument is not connected to a data stream, its icon in the Scene Navigator has a question mark.

3 If you have selected a Multistate instrument, you can specify its states.

1. Click the Browse button of the States property to open the States dialog.
2. To add a state, click Add in the States dialog.

3. Specify the properties of the state. For details on the properties, refer to States Dialog on page 202.



4 To add more instruments to the scene, repeat the last steps.

5 When instruments overlap, you can specify their layer. In the Scene Navigator, select an instrument under the assigned observer and choose one of the following commands or click the appropriate button in the Observation – Layer Instrument ribbon group:

- Bring to Front
- Move Layer Down
- Move Layer Up
- Send to Back

6 To hide an instrument, open the context menu of the instrument in the Scene Navigator and clear Visible.

7 To delete an instrument, open the context menu of the instrument in the Scene Navigator and choose Delete.

Result

The instruments display the values of the connected simulation variables in the animation.

Related topics

Basics

Basics on Using Instruments in the Scene 31

References

Bring to Front 143
Create (Instrument) 144
Create Bar Instrument 146
Create Gauge Instrument 148
Create LED Instrument 149
Create Numeric Instrument 149
Create Picture Instrument 151
Create Text Instrument 152
Delete (Instrument/Instrument Panel) 154
Move Layer Down 159

Basics on Using Instrument Panels in the Scene

Introduction

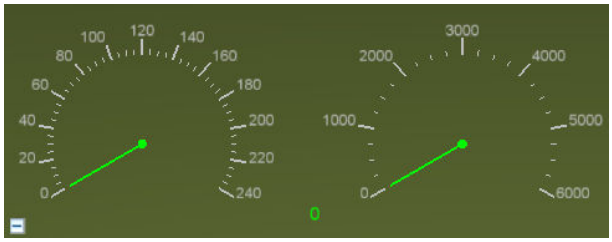
The purpose of the Instrument Panels is to display several instruments at once.

Instrument Panels

Instrument Panels can contain several instruments of different types, refer to Basics on Using Instruments in the Scene on page 31. When instruments are used on Instrument Panels, they become child instruments. Child instruments have the same properties as stand-alone instruments except for the properties for positioning. The positions of child instruments are specified relative to the Instrument Panels.

You can create child instruments directly on Instrument Panels or move existing stand-alone instruments to them.

The following illustration shows an Instrument Panel with two Gauge instruments and one Numeric instrument.

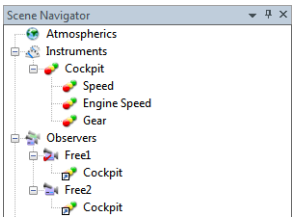


The Instrument Panels have the following features:

- You can specify several properties of the Instrument Panels, such as position, visibility, background color, and background pictures.
- You can specify the Instrument Panel's positions can be specified in pixels or as a percentage value of the view.
- You can attach Instrument Panels to movable objects or give them a static position in the 3-D View.
- Instrument Panels are linked to one or more observers.
- The Instrument Panel's size is specified by the size and position of the instruments which it contains.
- The position of the child instruments are specified in pixels.
- Instrument Panels can be minimized so that only their title texts are displayed: 

- The title text can display various details of the connected signal data of the primary child instrument.

- The Scene Navigator lists the Instruments Panel under the Instruments folder. Below the Instrument Panels, the child instruments are listed.



Limitations when using Instrument Panels

MotionDesk's instruments do not support script fonts, such as the *Vladimir Script* font. You can identify script fonts by their name.

Related topics

Basics

Basics on Using Instruments in the Scene 31

HowTos

How to Use Instrument Panels in the Scene 37

References

Create Panel 150
Instrument Properties 192

How to Use Instrument Panels in the Scene

Objective

You can use Instrument Panels in the scene to use several instruments at once.

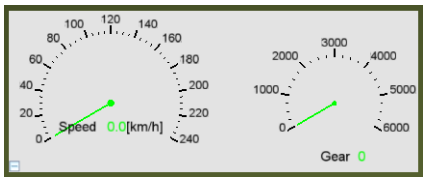
Size of an Instrument Panel

The form of an Instrument Panel is rectangular. It is a bounding box that includes the child instruments. The size of an Instrument Panel results from the positions and sizes of the child instruments.

When you add an instrument to the Instrument Panel, the Instrument Panel's size can change.

- If the Instrument Panel is static (Attached to property is static, it is not attached to a movable object), the Instrument Panel's position and size is recalculated automatically to include the new child instrument. The position of the child instrument is not changed.
- If the Instrument Panel is attached to a movable object, the position of the child instrument is set to 0, 0 (X-Offset to Frame = 0 and Y-Offset to Frame = 0).

The following illustration displays an Instrument Panel containing 3 child instruments. The background color is gray to visualize the size of the Instrument Panel.



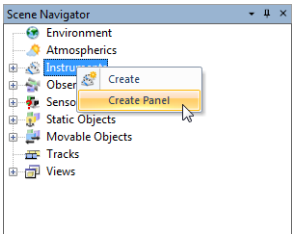
Specifying instruments

For information on how to specify the instruments of an Instrument Panel, refer to How to Use Instruments in the Scene on page 33.

Method

To use Instrument Panels in the scene

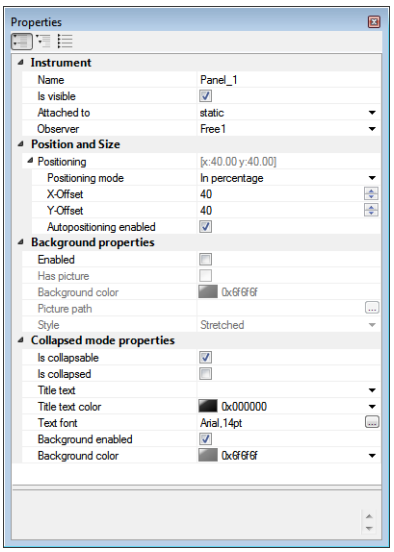
1 In the Scene Navigator, open the context menu of the Instruments folder and choose Create Panel.



Or

On the Observation ribbon, click Create Instrument – Panel.

MotionDesk creates a new Instrument Panel and displays its properties in the Properties pane.



2 In the Properties pane, specify the properties of the Instrument Panel. For details on the properties, refer to Instrument Properties on page 192.

3 To add an existing instrument to the Instrument Panel, you have two possibilities:
   - Drag the instrument to the Instrument Panel in the **Scene Navigator**.
   - Open the context menu of the instrument in the **Scene Navigator** and choose **Add to Panel**.

     The **Select Panel** dialog opens for you to let you choose the Instrument Panel.

   The instrument is moved to the Instrument Panel.

4 To add an new instrument to the Instrument Panel, open the Instrument Panel's context menu in the **Scene Navigator** and choose **Create Instrument**.

   A new instrument is added to the Instrument Panel.

5 Click the instrument that you added before. In the **Properties** pane, specify its properties. For details on the properties, refer to Instrument Properties on page 192.

6 To add more instruments to the Instrument Panel, repeat the last steps.

7 When child instruments overlap, it is useful to specify their layer. In the **Scene Navigator**, select a child instrument of the Instrument Panel and choose one of the following commands or click the appropriate button in the **Observation – Layer Instrument** ribbon group.

    Bring to Front
    Move Layer Down
    Move Layer Up
    Send to Back

8 To remove a child instrument from the Instrument Panel, you have two possibilities:
   - Drag the child instrument to the **Instruments** node in the **Scene Navigator**.
   - Open the context menu of the child instrument in the **Scene Navigator** and choose **Remove from Panel**.

   The instrument is moved to the Instruments node in the **Scene Navigator**. Its position in the scene is not changed.

9 To minimize an Instrument Panel, press the **Shift** key and click the minus sign

   or select the **Is collapsed** property in the **Properties** pane.

10 To maximize an Instrument Panel, press the **Shift** key and click the plus sign

   Or

   Clear the **Is collapsed** property in the **Properties** pane.

11 To hide an Instrument Panel, open its context menu in the **Scene Navigator** and clear **Visible**

Or

Select the **Is visible** property in the **Properties** pane.

12 To delete an Instrument Panel, open its context menu in the **Scene Navigator** and choose **Delete**.

Result

You created an Instrument Panel containing instruments.

Related topics

Basics

Basics on Using Instrument Panels in the Scene 36

References

Add to Panel 140
Bring to Front 143
Create (Instrument) 144
Create Panel 150
Delete (Instrument/Instrument Panel) 154
Move Layer Down 159
Move Layer Up 159
Remove from Panel 161
Send to Back 164
Visible (Instrument/Instrument Panel) 169

How to Configure the Title Bar of a Minimized Instrument Panel

Objective

The title bar of an Instrument Panel can display ordinary text and various details of the connected signal data of the primary child instrument.

Primary child instrument

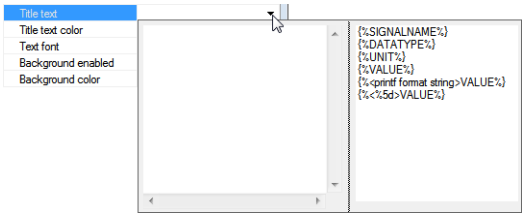
You can select one child instrument of an Instrument Panel as the primary child instrument. The title text of a minimized Instrument Panel can be configured so that it contains the signal name, unit, data type, and value of the signal that is assigned to the primary child instrument.

The child instrument that you create first on an Instrument Panel, is the primary child instrument by default. However, you can specify another child instrument as the primary child instrument.

Method

To configure the title bar of a minimized Instrument Panel

1. To specify the primary child instrument, open the context menu of an instrument of the Instrument Panel in the Scene Navigator and choose Make Primary.
2. In the Scene Navigator, select the Instrument Panel.
   The Properties pane displays its properties.
3. In the Title text property, click the arrow to open an editor.



4. Enter the text. You can use the following macros for displaying the signal.

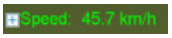
Macro	Description
`{%SIGNALNAME%}`	Name of the signal as specified in the simulation model.[1)]
`{%DATATYPE%}`	Data type of the signal as specified in the simulation model.[1)]
`{%UNIT%}`	Unit of the signal as specified in the simulation model.[1)]
`{<printf format string>%VALUE%}`	Value of the signal. You can optionally specify a printf format string to format the output of the value. For example, to output an integer value with at least 5 digits, enter: `{%<%5d>VALUE%}`.

1) Refer to MD_Instrumentation (MotionDesk Calculating and Streaming Motion Data 🕮).

Example

To display the signal name, value, and unit, enter the following string:

```
{%SIGNALNAME%}: {%<%5.1f>VALUE%} {%UNIT%}
```

When the primary child instrument is assigned to a Speed data stream, the minimized Instrument Panel looks like this: 

Result

When the Instrument Panel is minimized, its title bar displays the signal of the primary child instrument.

Related topics

References

Make Primary..... 158
MD_Instrumentation (MotionDesk Calculating and Streaming Motion Data 🕮)

Using State Objects and Animated Characters in the Scene

Introduction

You can use state objects to visualize states of objects in accordance with the simulation variables. State objects are special 3-D objects that have subobjects whose properties or appearance can change during animation.

Where to go from here

Information in this section

Basics of Using State Objects in the Scene

Introduction

You can use state objects in a scene to visualize various states of 3-D objects.

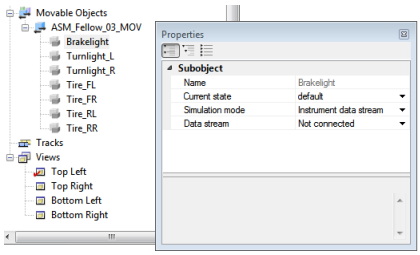
The standard library of MotionDesk lets you use state objects, such as a chassis with brake lights and turn lights that can be either dark or bright, wheels that can rotate without any data from the simulation, and gantries that can display different traffic signs.

An additional license for MotionDesk lets you use a library containing animated characters, such as humans or animals, which can move their extremities to visualize different movements, such as standing, walking, or running.

State objects

State objects are special 3-D objects which have subobjects whose look can change according to values they receive from a data stream. This feature is used to visualize the active lighting of brake lights and turn lights on 3-D chassis objects or the rotation of the tires. The active lighting of such 3-D objects is simulated by exchanging the color of the subobject. See the following example.

You can identify state objects in the **Scene Navigator**. The state objects have subobjects that are visible as children in the tree. The subobjects have properties which are listed in the **Properties** pane. See the following example.



In the subobject's properties you can select the signal of the data stream that contains the state values for the subobject.

Setting the lights of state objects

All the lights of a state object can be set by one signal of a data stream. The signal is transmitted like a signal for an instrument using an **MD_Instrumentation** block in the simulation model. The signal must have the uint32 or uint64 data type. To get the state of each subobject, the signal is evaluated bitwise. The following table shows which subobjects of a chassis object are assigned to the various bit numbers.

Bit numbers:	32 … 13	12 … 9	8 … 5	4 … 1
Subobject:	Not used	Turn light right	Turn light left	Brake light

To set the states of the subobjects, set the corresponding bits according to the values in the following table.

Subobject	Bit Numbers	State	Bit Values	Decimal Value
Brake light	4 … 1	Off	0 0 0 0	0
		On	1 1 1 1	15
Turn light left	8 … 5	Off	0 0 0 0	0
		Steady on	0 0 0 1	16
		Flashing on	1 1 1 1	240

Subobject	Bit Numbers	State	Bit Values	Decimal Value
Turn light right	12 … 9	Off	0 0 0 0	0
		Steady on	0 0 0 1	256
		Flashing on	1 1 1 1	3840

The signal value to be transmitted must be the sum of the values for each subobject.

Example If the brake light of a vehicle should be on and its right turn light should be flashing, the signal must have the following value in binary format:

```
States = 0000 0000 0000 0000 0000 1111 0000 1111
```

Or in decimal format:

```
States = 3855
```

For information on how to implement the signal transmission, refer to How to Prepare the Real-Time Model for Using Instruments or State Objects (MotionDesk Calculating and Streaming Motion Data 🕮).

Rotation of the tires

In traffic scenarios, the movement of the ASM vehicle and further vehicles (fellows) are simulated. MotionDesk has state objects visualizing vehicles with their wheels which can rotate. For using these state objects it is sufficient to calculate the movement of the fellows in the simulation. The rotation of the fellow's tires can be calculated by MotionDesk in an internal simulation. This reduces the amount of data that must be transferred from the simulation to MotionDesk.

Related topics

References

MD_Instrumentation (MotionDesk Calculating and Streaming Motion Data 🕮)
Subobject Properties (MotionDesk Scene Creation 🕮)

How to Use State Objects in MotionDesk

Objective

A state object is configured like any other 3-D object. However, you must also specify the properties of its subobjects.

Restrictions

There are some restrictions in the current MotionDesk version:

- Only state objects in a chassis with brake lights and turn lights are shipped.
- The 3-D objects in the following folders are not state objects:
  - `Truck` folder
  - `Trailer` folder

- `Car_Pickup` folder
- `Car_Formula1` folder

- User-defined state objects are not supported.

Preconditions

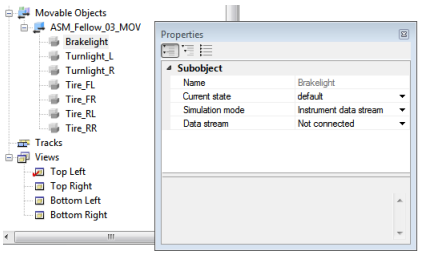
- A project must have been created and an experiment must be active.
- The real-time application can be prepared to calculate signal values for the state objects. An instrument data stream delivers a signal which sets the state. Motion data can be assigned to move the state object like any other movable object. Refer to How to Prepare the Real-Time Model for Using Instruments or State Objects (MotionDesk Calculating and Streaming Motion Data 🕮).

Method

To use state objects in MotionDesk

1. In the Library Browser, choose a state object.
2. Drag the state object into the 3-D View.

   The state object can be configured like any other 3-D object. However, you must also specify the properties of its subobjects.
3. In the Scene Navigator, select a subobject of the state object and specify its properties. For details on the properties, refer to Subobject Properties (MotionDesk Scene Creation 🕮).



4. Repeat the last step for all the other subobjects of the state object.
5. Repeat the previous steps for all the other state objects.

Result

You have inserted state objects.

Related topics

References

Subobject Properties (MotionDesk Scene Creation 🕮)

Basics of Using Animated Characters in the Scene

Introduction

You can use animated characters in a scene to visualize traffic participants that move in the scene.

Required license

To use animated characters in MotionDesk, a special license (MotionDesk Animated Characters) is required.

Animated characters

Animated characters are human and animal 3-D objects that can move their extremities autonomously, see the following example.

Properties specify the kind of movement, such as standing, walking, or running. Further properties specify the appearance, clothing, or ethical look. You can specify the properties manually in the **Properties** pane or assign a data stream from the simulation.

The following table shows two examples of animated characters and their possible settings.

Object	Subobject	Possible State
Horse	Animation	Current state specifies the kind of motion. Values: idle, stand, walk, run
	Appearance	Current state specifies the appearance. Values: horse_black, horse_white, horse_brown
Female_Casual_1	Animation	Current state specifies the kind of motion. Values: idle, stand, walk, stroll, walk fast, jog, run, passenger
	Clothing	Current state specifies the clothing. Values: casual_1, …, casual_9
	Ethnical Look	Current state specifies the ethical look. Values: white, black, asian, hispanic, white2

Moving animated characters

Like other movable objects, you can assign motion data to the animated characters. Therefore it is possible to control the movement of the objects in the scene and their animation by the simulation model.

The movement of animated characters is synchronized with ASM maneuver time. Therefore, the speed of the movements of the animated characters changes with the changes in the simulation, for example, the speed of a person walking, running, or cycling or the movement of an animal is relative to the speed of the simulation. This ensures a realistic determination of sensor data in a sensor simulation. If a simulation is stopped, the animated characters also stop moving.

To work with animated characters, you must have the relevant MotionDesk license. For more information, refer to MotionDesk Licenses (MotionDesk Basics 🕮).

Related topics

HowTos

How to Use Animated Characters in MotionDesk..... 48

References

MD_Instrumentation (MotionDesk Calculating and Streaming Motion Data 🕮)
Subobject Properties (MotionDesk Scene Creation 🕮)

How to Use Animated Characters in MotionDesk

Objective

An animated character is configured like any other 3-D object. However, you must also specify the properties of its subobjects.

Preconditions

- A project must have been created and an experiment must be active.
- If the real-time application is to control the animated character, the application must be extended. An instrument data stream can deliver a signal which sets the state. Motion data can be used to move the animated character like any other movable object. Refer to How to Prepare the Real-Time Model for Using Instruments or State Objects (MotionDesk Calculating and Streaming Motion Data 🕮).

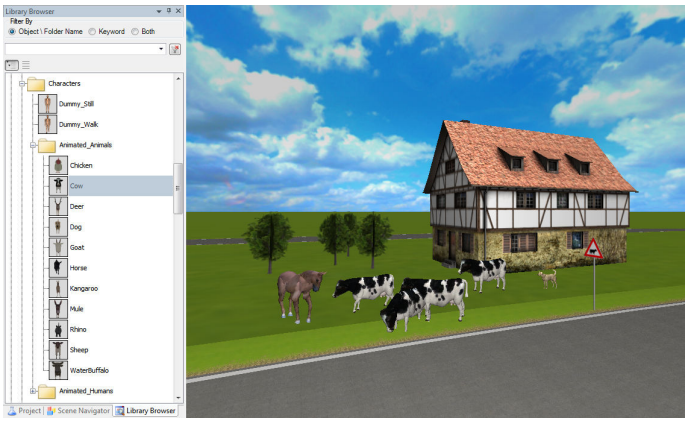
  The movements of the animated characters are synchronized with the speed of the simulation maneuvers

Method

To use animated characters in MotionDesk

1 In the Library Browser, open the Character/Animated_Animals or Character/Animated_Humans folder and choose a 3-D object.

2 Drag the 3-D object into the 3-D View.



The animated character can be configured like any other 3-D object. However, you must also specify the properties of its subobjects.

3 In the Scene Navigator, select a subobject of the animated character and specify its properties. For details on the properties, refer to Subobject Properties (MotionDesk Scene Creation 🕮).

4 Repeat the last step for all the other subobjects of the animated character.

5 Repeat the previous steps for other animated characters.

Result

You have inserted animated characters.

Related topics

Basics

Basics of Using Animated Characters in the Scene.....47

Starting an Animation

Introduction

When the real-time application runs, you can start the animation in MotionDesk.

Where to go from here

Information in this section

Information in other sections

1-PC Solution (MotionDesk Basics 🕮)
Introduces visualization with real-time hardware and one PC.

Multi-PC Solution (MotionDesk Basics 🕮)
You can use several PCs for visualizing the same scene from different point of views.

Starting the Simulation

Preconditions

Before you can start an animation, the real-time application must be started. The start procedure for the real-time application depends on the hardware system used. For information on the hardware system, refer to System Overview (MotionDesk Basics 🕮).

Using a 1-PC solution

If you work with a 1-PC solution, ControlDesk and MotionDesk run on the same PC. Use ControlDesk to download and start the real-time application to the real-time system. For instructions, refer to Handling Real-Time and Offline Simulation Applications (ControlDesk Platform Management 🕮).

Using a multi-PC solution

If you work with a multi-PC solution, ControlDesk runs on a host PC and MotionDesk runs on the visualization PCs. Use ControlDesk to download and start the real-time application. For instructions, refer to the Handling Real-Time and Offline Simulation Applications (ControlDesk Platform Management 🕮). Use one of the MotionDesk PCs to start the animation. For instructions on the installation of a multi-PC solution, refer to Setting Up a Multi-PC Solution (MotionDesk Calculating and Streaming Motion Data 🕮).

Simulink simulation

If you work with a PC-based Simulink simulation, Simulink and MotionDesk run on 1 PC or 2 separate PCs. In both cases, Simulink is used to start the simulation and MotionDesk to start the animation. For instructions on installation, refer to Setting Up Visualization for a Simulink Simulation (MotionDesk Calculating and Streaming Motion Data 🕮).

Related topics

Basics

1-PC Solution (MotionDesk Basics 🕮)
Multi-PC Solution (MotionDesk Basics 🕮)

How to Start the Animation

Objective

When the motion data are streamed to MotionDesk, the animation can be started.

Preconditions

Before starting the animation, ensure that the scene is prepared. Movable objects, instruments, and state objects should be connected to data streams. Refer to the following topics:

- Setting up Movable Objects on page 24
- Using Instruments in the Scene on page 31
- Using State Objects and Animated Characters in the Scene on page 43

Method

To start the animation

1 On the Home ribbon, click Simulation – Go Online.

Result

If the scene is prepared properly, the animation runs. Otherwise, check the settings of the scene.

In a multi-PC solution, animation start and stop are synchronized on all connected MotionDesk PCs, regardless of which PC the command is executed from.

When MotionDesk displays the animation, the motion data is written to a frame buffer. The buffer can then be stored for replay, see How to Store Motion Data on page 99.

Related topics

References

Go Online 182

How to Set the Rendering and Lighting Levels

Objective

You can set the rendering compression levels and select the advanced light level mode for a more realistic scene.

Rendering

Texture compression levels You can select which textures are compressed during rendering. You can select uncompressed to render all textures, minimum compression to only render textures that require color, or maximum compression to render no textures.

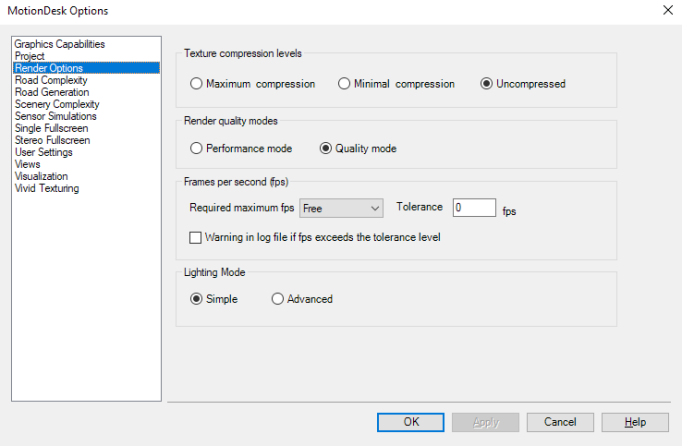
Render quality mode You can select the render mode for improved performance.

Lighting mode You can enable advanced lighting mode to add additional light objects to provide more realistic lighting in the scene.

Method

To select the rendering options

1 On the File ribbon, click Options to open the MotionDesk Options dialog and change to the Render Options page.



2 In Texture compression levels, select Maximum, Minimal, or Uncompressed (none).

3 In Render quality modes, select Performance mode, or Quality mode.

4 In Lighting mode, select Simple, or Advanced.

5 Select OK to save your changes.

Result

The rendering options are specified and saved.

Related topics

References

Render Options Page..... 208

How to Limit the Frame Rate

Objective

You can specify a maximum value for the frame rate.

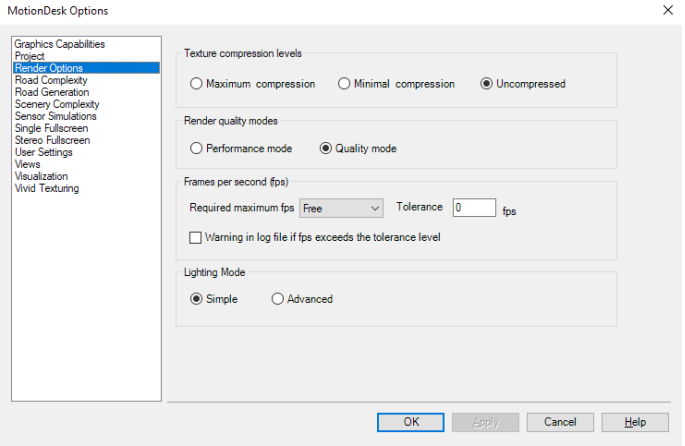
Maximum frame rate

You can specify the maximum number of frames per second for the animation in MotionDesk. The smaller this value, the more calculation power is available for other processes running on the PC.

Method

To select the maximum frame rate

1 On the File ribbon, click Options to open the MotionDesk Options dialog and change to the Render Options page.



2 In Required maximum, select the maximum limit number of frames per second. You can select a value in frames per seconds or "free". If you select "free", MotionDesk captures all processing time which is not required by other processes for the animation. Otherwise the frame rate is limited by the selected maximum value.

3 If you select a maximum value, you can enable a warning message that occurs when the frame rate is below the maximum value and a tolerance value. If the warning message is output and then the frame rate reaches the specified maximum value again, an information message is output.

Result

The frame rate is limited.

Related topics

References

Render Options Page..... 208

How to Configure the 3-DView

Introduction

The MotionDesk display shows the animated scene. You can split the 3-D View into four windows and display different observer views of the animation. You can also apply image exposure and color filters for each view.

Where to go from here

Information in this section

How to Configure the 3-D Views

Objective

To observe the simulation from different viewpoints at the same time, you can display up to 4 windows in the 3-D View and change each window's proportions.

Views

MotionDesk has four views. You can display one of the views in the 3-D View or split the viewer to display multiple view windows.

For each of the views, you can select observer to display in the window and adjust the viewing angles.

You can also apply a number of filters to improve the detail of the objects in the scene when moving around the scene or when animating a scene in a simulation. For example, you can change the exposure settings and enable tone mapping, material view display, and pseudo coloring.

Method

To configure the views

1. On the Observation ribbon, click Views and the desired view arrangement.

   Or

   On the Home ribbon, click Views and the desired view arrangement.

   For example, select to display all four view window.
2. In the Scene Navigator, select Views and click on one of the four views.
3. In the properties pane, select the Observer to display scene for the selected observer in the view window.
4. Apply any exposure, tone mapping, material view, and pseudo coloring settings to analyze the objects in the scene.
5. Click a one of the four view windows.

   A red box is shown around the selected view window and a red tick is shown on the View in the Scene Navigator.
6. On the Home - Views click the single view window icon.

   The selected view in the previous step is displayed in this view window.

Tip

When displaying multiple view windows, you can double-click a view window to display only this view.

Result

You selected an observer for a view in the 3-D View and displayed single and multiple views.

What's next

You will learn to change the size of each 3-D View window. Refer to How to Adjust the 3-D View Windows on page 57.

Related topics

HowTos

How to Adjust the 3-D View Windows..... 57

References

3-D View Properties..... 205

How to Adjust the 3-D View Windows

Objective

To observe the simulation from different viewpoints, you can split the 3-D View into up to 4 windows and change the size of each window.

Method

To adjust the windows of the 3-D View

1 On the Observation ribbon, click Views and the desired view arrangement.

Or

On the Home ribbon, click Views and the desired view arrangement.

2 Move the mouse pointer to the border between two windows.

It changes to one of the following symbols:

3 Drag the mouse until the desired proportions are reached.

4 Double click one of the views in the 3-D View to display only this view in the 3-D View in maximum size.

5 To display the scene on full screen,

Go to the View ribbon and click Full Screen – Single Full Screen.

Or

Go to the Home ribbon and click Views – Single Full Screen.

To exit the full screen mode, press **Esc**.

Result

You changed the view of the scene displayed in the 3-D View.

What's next

You will select the Material ID Preview in the 3-D View. Refer to How to Display the Material ID Preview in MotionDesk on page 60.

Related topics

HowTos

How to Configure the 3-D Views....55

References

Always on Top....141
Four Views....157
One View....161
Single Full Screen....165
Tile Horizontally....166
Tile Vertically....167

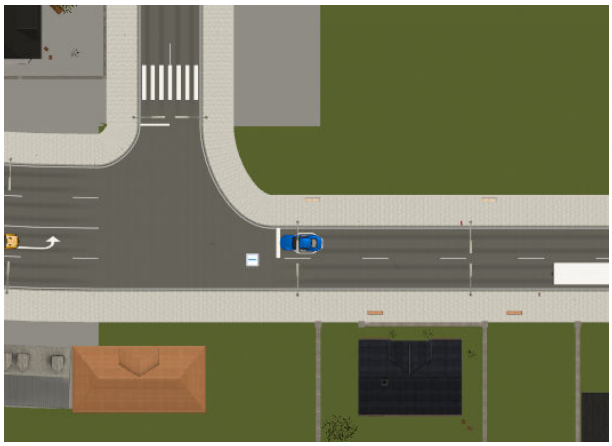
How to Use the Bird's-eye View

Objective

To enable a bird's-eye view above the MotionDesk scene.

Overview

For each of the four available views of the MotionDesk 3-D View, you can enable the Bird's-Eye View. The bird's-eye view is a two-dimensional view above the scene. An example is as follows.



You can roll the mouse wheel button to increase the area of the ground viewable in the bird's-eye view. You can also specify the bird's-eye view manually in the properties. This has the effect of zooming in and out above the scene.

The height and width properties of the bird's-eye view are the rectangular dimensions in meters on the ground. Therefore, if you roll the mouse wheel or change the properties, you change the size in meters of the ground area that is visible in the bird's-eye view. The aspect ratio of the view is the same, for example, if you change the width property of the bird's-eye view, the height property automatically changes.

With unlocked or free observers, you can also freely move around the scene in the bird's-eye view. You can click and hold the left mouse button and drag the mouse in any direction to move over the scene. If the observer is locked, the view returns to the original position, when you release the mouse button.

If an observer is fixed to a movable object in the selected view, the bird's-eye view follows the object through the scene.

Note

The 2-D view is no longer supported and cannot be selected in View - Switch Controlbars.

Method

To use the bird's-eye view

1. In the MotionDesk 3-D View, click on one of the views.
2. In the Scene Navigator, click Views and select the active view. The active view is the view with a red tick icon.

   The Properties pane is displayed.
3. In the Bird's-Eye View properties, select Enabled.

   The 3-D View displays a view above the scene.
4. Roll the mouse wheel backward to increase the width and height of the ground area in the scene that is viewable in the bird's-eye view.

   This has the effect of zooming out.
5. Roll the mouse wheel forward to decrease the width and height of the ground area in the scene that is viewable in the bird's-eye view.

   This has the effect of zooming in.
6. Change the Width property to a higher value.

   The height of the ground area visible to the bird's-eye increases. The Height property also changes.

   This has the effect of zooming out.
7. Click and hold the left mouse button and move around over the scene in the bird's-eye view. Release the mouse button.

   If the observer is locked, the bird's-eye view returns to the original position.
8. If you have a movable object in the scene, in the View Properties, select an Observer that has the Fixed to object behavior and is attached to the movable object, for example, a car chassis.

   The bird's-eye view moves to the position of this observer.
9. Start the simulation, or click play on the Motion Player to start the animation.

   The 3-D view displays the bird's-eye view and follows the vehicle through the scene.

Result

You enabled the **Bird's-Eye View** in a 3-D view to display a two-dimensional view from above the MotionDesk scene. You also followed a vehicle in the simulation in the bird's-eye view.

Related topics

Basics

Navigating Through the Scene with Observers 85
Working with Observers 79

HowTos

How to Assign Motion Data to Movable Objects 26

References

3-D View Properties 205

How to Display the Material ID Preview in MotionDesk

Introduction

You can select the material ID preview in the MotionDesk view properties to display the preview colors of the materials that are assigned to the pixels of the 3-D object images in the scene.

Overview of the material preview

For simulation with sensors, you can add materials and assign them to the pixels of the 3-D objects that are used in the MotionDesk scenes.

The materials and their properties, including a preview color are maintained in the `materialmapping.xml` material database file, which can be maintained using the Material Database Editor.

Most of the 3-D objects in the library are saved in graphics texture files which contain the mapping of the materials in the database to the pixels of the object images.

In the MotionDesk **View** properties, you can select the material ID preview of the scene. Each object in the scene is displayed with the preview colors of the materials that are assigned to the pixels of the object images.

Note

If an object material is transparent, the material ID preview color for this material is displayed. The materials behind the transparent glass are not shown. For example, a glass windshield is shown instead of the steering wheel behind window.

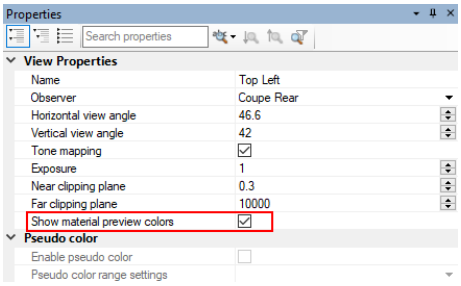
For more information on merging the material database, refer to How to Merge the Material Database (MotionDesk Sensor Simulation Control 🕮).

Method

To display the material ID preview colors in the MotionDesk observer view

1 In MotionDesk, select a camera sensor or a fish-eye sensor in the Scene Navigator.

2 In the Properties pane, select View Properties - Show material preview colors.



Note

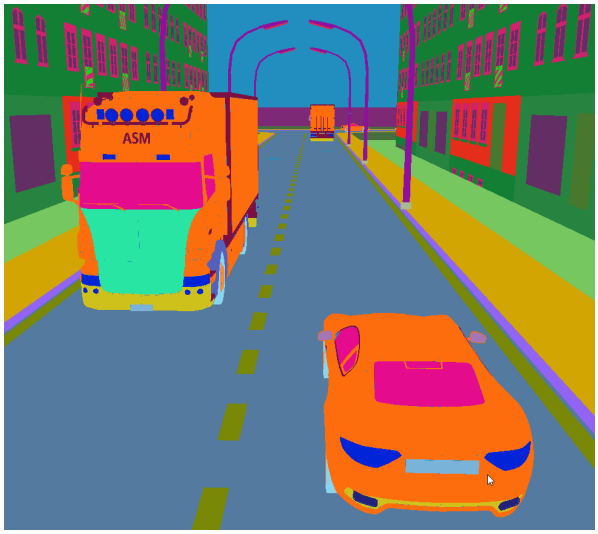
You can select Show material preview colors only if Enable pseudo color is not selected.

3 Move around the 3-D View to view the objects in the scene.

Each material for each of the objects in the MotionDesk are displayed in color. The color of each material is unique and is specified in the material mapping XML.

Without material preview colors

Material preview colors selected



Result

You selected Show material preview colors in the MotionDesk View properties and viewed the observer view that displays the material ID preview colors for each object in the scene.

What's next

You will learn to work with pseudo color rendering in the 3-D View. Refer to Working with Pseudo Coloring in the MotionDesk 3-D View on page 62.

Related topics

Basics

Basics on Material Management (Sensor Simulation Manual 🕮)
Working with Material IDs in Postprocessing Camera and Fish-Eye Output (MotionDesk Sensor Simulation Control 🕮)

References

Material Database (Sensor Simulation Manual 🕮)

Working with Pseudo Coloring in the MotionDesk 3-D View

Introduction

In MotionDesk, you can apply pseudo coloring to each of the observer views to analyze the scene using the configured light distribution and intensities.

Overview

The 3-D View displays the MotionDesk scene in a window. It can be divided into up to 4 subwindows and each sub-window can be configured to display the scene from a different perspective. The views are selected in the ribbon or in the Scene Navigator.

You can also enable a two-dimensional birds's-eye view of the scene from above using the properties pane.

The 3-D View can also be displayed in full screen mode on one or more screens. The selected view is shown in the full screen.

In MotionDesk, you can enable pseudo colors for each of the observer views in the 3-D View that display scene. You can also configure the pseudo color ranges.

The selected color range is displayed in the view based on the brightness intensity of the surface the object.

The pseudo colors allows the user to analyze the scene using the configured the light distribution and intensities in the scene. Pseudo coloring can improve the detail in the scene to display objects that are not visible due to the lack of light. These might be objects that the camera and fish-eye sensors in sensor simulation have detected or missed.

Pseudo coloring in MotionDesk and sensor simulation

In a real world ADAS/AS scenario, sensors can detect objects in the scene inside their clipping point and field of view settings. They might detect objects in the scene that are not displayed in the MotionDesk scene. For example, the camera and fish-eye sensors can be configured to encode the image with the 24-bit bayer pattern. Therefore dark areas of a scene can be shown as black in the observer view but objects can still be present and visible for the sensors. You can enable and configure pseudo colors to help display the darker areas of the scene in the observer view.

Note

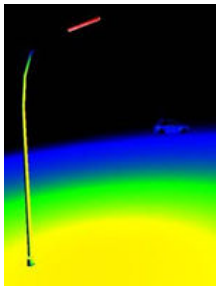
In the 24 bit bayer pattern, only one color channel instead of three is stored for each pixel. The single channel contains more details.
Pseudo coloring allows the user to analyze this high dynamic range feature of camera and fish-eye sensors.
For more information on the camera and fish-eye sensor output settings, refer to Basics on the Camera and Fish-Eye Output Settings (MotionDesk Sensor Simulation Control 🕮).

You can use the pseudo coloring to map the brightness of the objects in the scene to colors from a color scale. You can then analyze the objects in the scene to investigate if the sensor has correctly identified all objects that are present. This includes objects that are hidden in dark areas of the scene observer view.

You might also decide that the object detection of the ECU should be improved if a sensor does not detect an object it should have. For example, the sensors

might miss objects in low light conditions or in situations where too much light from the sun or street lighting is reflected off objects in the scene causing glare.

In the following example, if pseudo coloring is selected, the vehicle object in the scene becomes visible in the observer view. This might have been detected by the sensor but it was not visible in the scene to the user.

Without pseudo coloring	Pseudo coloring selected
	

Note

The pseudo colors are shown in the 3-D View and in full screen mode. They are not shown in the sensor composition display of the scene in sensor simulation.

Pseudo coloring scenarios

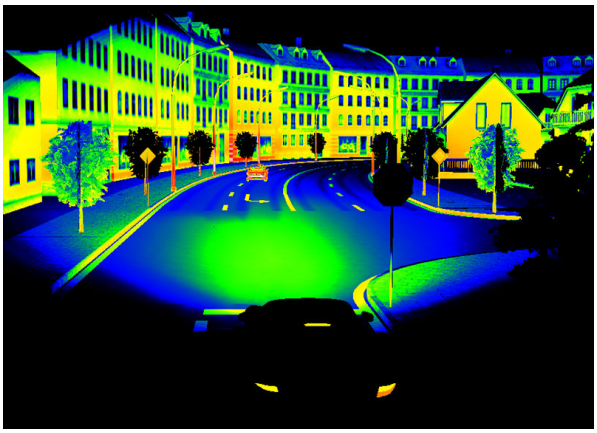
Low light scenario You can select the night mode in the MotionDesk scene atmospherics to display a low light scene.

For more information on Atmospherics, refer to Atmospheric Settings on page 128.

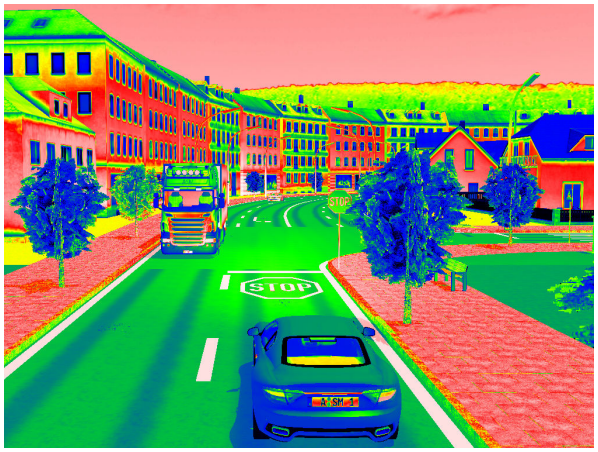
In the following example, the ego-vehicle has light objects and produces the only source of light in the night scene. Certain objects in the scene that are visible to the sensors in sensor simulation are not be displayed in the observer view.

With pseudo coloring, you can see the oncoming vehicle on the road and the buildings in the background.

Another low light example is where the ego-vehicle in the simulation is in a tunnel where the tunnel lamps are the only light source.

Without pseudo coloring	With pseudo coloring
	

In the following daylight image, the light intensities reflected of the buildings and the vehicles in the scene are represented by the colors from the color range spectrum. You can configure the color intensities for the black, blue, green yellow, red and white colors in the color range settings.



Related topics

Basics

Basics on the Camera and Fish-Eye Output Settings (MotionDesk Sensor Simulation Control 🕮)
User Interface of MotionDesk (MotionDesk Basics 🕮)

HowTos

How to Configure Scene Pseudo Coloring in MotionDesk..... 66

References

3-D View Properties..... 205

How to Configure Scene Pseudo Coloring in MotionDesk

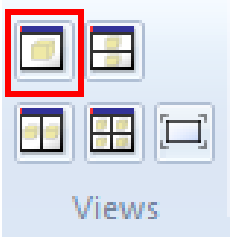
Introduction

In MotionDesk, you can enable and configure the pseudo coloring for each of the observer views.

Method

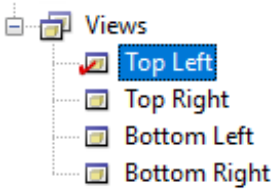
To configure MotionDesk scene pseudo coloring

1 In MotionDesk, select the Home ribbon. In Views, select the single display window for the 3-D View.

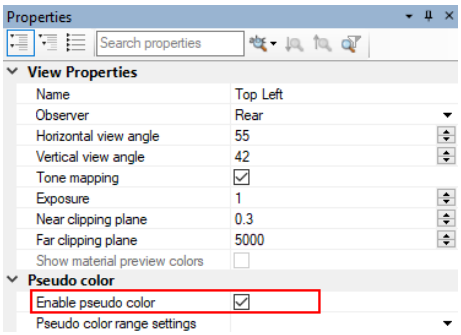


The active view is displayed in one screen in the 3-D View.

In the Scene Navigator the active view is shown with a red tick.

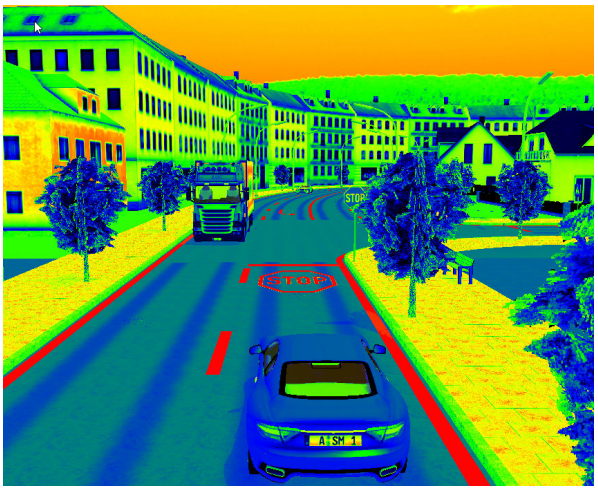
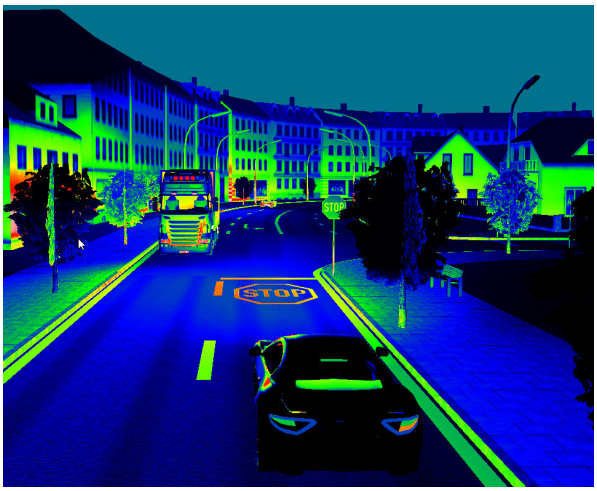


2 In the Properties pane, select Pseudo color - Enable pseudo color.



The 3-D View is displayed with pseudo coloring with the default color range as follows:

The colors displayed for each object in the scene are based on the brightness intensity of the surface the object.

Pseudo coloring - Day time atmospherics	Pseudo coloring - Night with vehicle light objects
	

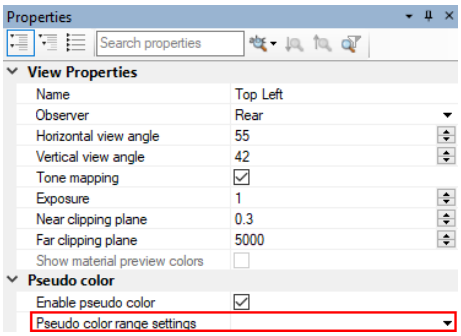
Note

To display the lights in pseudo coloring mode, you must use the default luminance distribution map for the light objects. Light objects can be added to the vehicles in advanced lighting mode.

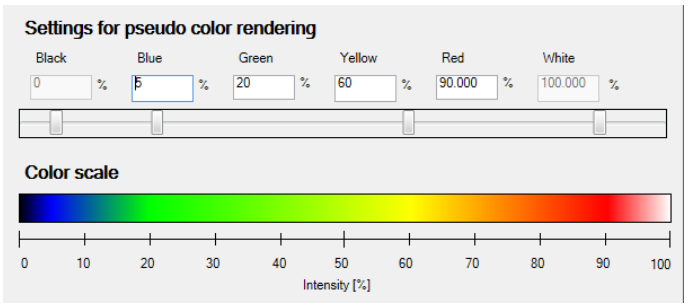
- Low beam: <MotionDesk_Root_Dir> /Assets/Textures/LowBeam.tga
- High beam: <MotionDesk_Root_Dir> /Assets/Textures/HighBeam.tga

For more information on light objects, refer to Basics of Light Objects (MotionDesk Scene Creation).

3 In the Properties pane, select Pseudo color range settings.



The Settings for pseudo color rendering dialog opens as shown in the following image:



4 Change the color range settings individually by changing the percentages in the color fields. You cannot select values beyond the boundaries of the preceding or next color in the scale.

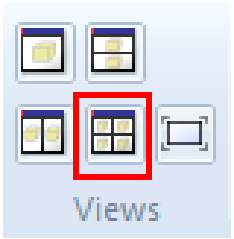
Tip

You can also move the sliders with the mouse or keyboard arrow keys to adjust the color intensity. The sliders for each color cannot be moved beyond the value of the preceding or next color in the scale.

The percentages in the fields, the slider positions and the color range scale are all adjusted in line with the changes you make.

The 3-D View also changes when you adjust the color range.

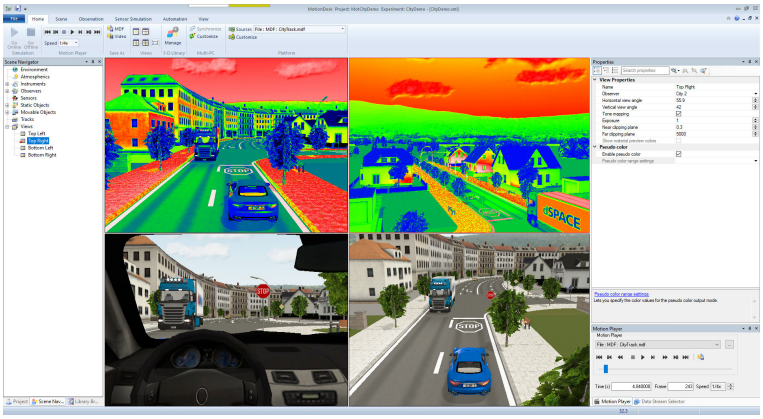
5 In MotionDesk, select the Home ribbon. In Views, select the quartered display window for the 3-D View.



The 3-D View shows four subwindows with different views of the scene.

6 In the Scene Navigator, select Views - Top Right and in the Properties pane, select Enable pseudo color.

The Top Right view is shown with pseudo coloring.



Tip

You can click on the Top Right window to put this window in focus.

Then select Home - Views - Full screen to display this pseudo colored 3-D view of the scene in full screen mode.

Result

You enabled and configured pseudo coloring on two of the four views. You also displayed a view in full screen mode.

The objects in a poorly illuminated part of the scene, for example, in a tunnel or in night mode are more clearly visible.

What's next

You will configure display filters in the 3-D View. Refer to How to Configure Tone Mapping and Exposure on page 70.

Related topics

Basics

User Interface of MotionDesk (MotionDesk Basics 🕮)

HowTos

Working with Pseudo Coloring in the MotionDesk 3-D View..... 62

References

3-D View Properties..... 205

How to Configure Tone Mapping and Exposure

Introduction

You can configure tone mapping and exposure scene image enhancements in the MotionDesk scene properties to improve the detail and light intensities of the objects in the 3-D View.

Overview

In the MotionDesk View properties, you can select to enhance the image detail for this view. You can select to enable tone mapping and adjust the exposure. The changes are applied to all objects and the background in the scene.

Tone mapping You can enable tone mapping for the observer view to improve the range of light intensities of the objects in the scene. This is helpful to improve the details of the objects in the image. Tone mapping reduces the contrast of objects with high contrast values for display on the PC screen. Images with a high dynamic range are compressed to be displayed on an output screen that supports a standard dynamic range.

Exposure You can adjust the amount of light exposure to be shown in the 3-D View for the observer. When you increase the exposure of the scene, the scene brightness increases. This is helpful in darker scenes where the detail in the scene is not clear for the user. You can also reduce the exposure if the scene is too bright.

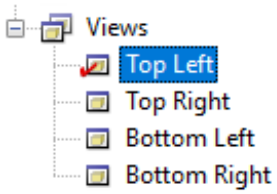
Note

Tone mapping and exposure settings apply to the observer view in the MotionDesk scene. They are not applied to the sensor image shown in the sensor composition window.

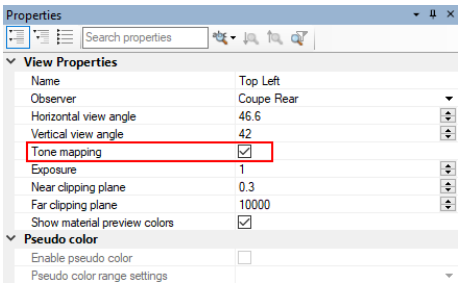
Method

To configure tone mapping and exposure

1 Select the active view in the Scene Navigator.

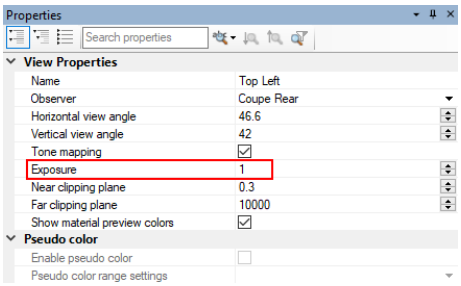


2 In the Properties pane, select View Properties - Tone mapping.



The image detail in the 3-D View is improved.

3 Click the up arrow button in View Properties - Exposure to increase the exposure in the 3-D View to 2. The default exposure is 1.



4 Click the down arrow button in View Properties - Exposure to reduce the exposure in the 3-D View to 0.5.

The 3-D View can appear as in the following example.

Exposure = 0.5	Exposure = 1	Exposure = 2
		

Result

You enabled tone mapping and configured the exposure settings for the 3-D View.

Related topics

References

3-D View Properties..... 205

How to Configure the MotionDesk Displays

Introduction

You can configure the MotionDesk animation to display in full screen and across multiple display screens

Where to go from here

Information in this section

How to Configure Full Screen Mode with Multiple Displays

Objective

If a MotionDesk PC has multiple screens, you can select observer views for each screen or spread a view across the screens in full screen mode.

Basics

You can select to automatically span the active visualization view in the 3-D View across all the connected available screens.

You can also manually configure which screens are used in full screen mode and which observer views are displayed in each screen.

- Automatically spread the active view across several screens. Refer to Part 1 on page 74.
- Manually select and order the displays used for full screen mode. Refer to Part 2 on page 75.
- Select specific views or observers for each display screen. Refer to Part 3 on page 75.

  Examples
  - Visualization views: Select the active view, one specific view, or all four to be shown in a grid.
  - Observer views: Select specific observer views.
- Activate the full screen mode in a simulation. Refer to Part 4 on page 76.

Note

If you selected the same screens for sensor simulation as for full screen mode, the sensor composition window is hidden behind the full screen display. You can press **Esc** to close full screen or stereo mode, or switch back to MotionDesk by pressing **Alt+Tab**.

For more information, refer to How to Configure a MotionDesk Screen for Sensor Simulation (MotionDesk Sensor Simulation Control 🕮).

Tip

In full screen or stereo mode, you can move around the 3-D View during the simulation or motion player playback. For more information, refer to Overview of the Observer Navigation Mode on page 85.

Preconditions

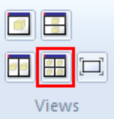
The MotionDesk PC must have several displays.

All displays must have the same resolution.

Part 1

To automatically spread the active view across several displays

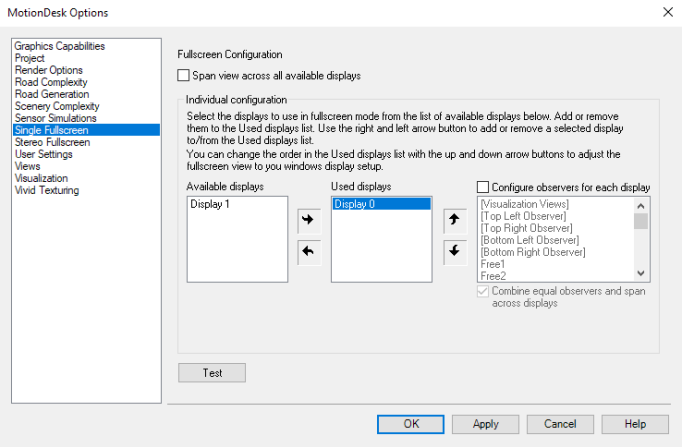
1 On the Home ribbon, select Views and click on the 4 view split screen icon.



2 Click on of of the four views in the 3-D View.

A red box highlights the select view. It is now the active visualization view.

3 On the File ribbon, click Options - Single Fullscreen.



4 Select Span view across all available displays.

The available display, used display, and the observer view selection lists are gray.

5 Click Test.
The selected active visualization view in the 3-D View is displayed in full screen mode on each of the used displays.

6 Click OK to close the dialog.

Interim result

You automatically spread the active view across several displays.

Part 2

To select the displays and the display order

1 On the File ribbon, click Options - Single Fullscreen.

2 Clear Span view across all available displays.

3 To add a screen for the full screen mode, select it in Available displays and click .

4 To remove a screen for the full screen mode, select it in Used displays and click .

5 To change the order of the screens, select a screen in Used displays and click or .

6 Click Test.
The active view in the 3-D View is displayed in full screen mode on each of the used displays.

Interim result

You select the displays for full screen mode and changed the display order.

Part 3

To configure specific views for each display

1 On the File ribbon, click Options - Single Fullscreen.

2 Clear Span view across all available displays.

3 Select a display in the Used displays list.

4 Select Configure observers for each display and select the observer to use for the selected display.
- [Visualization views]: This displays all four of the visualization views
  For more information on visualization views, refer to How to Configure the 3-D Views on page 55.
- [Top / Bottom / Left / Right Observer]: The observer that is configured in the specific visualization view is displayed. For example, select [Top Left Observer] to display the observer that is configured in this visualization view.

For more information on visualization views, refer to How to Configure the 3-D Views on page 55.

- To use an explicit observer, select it by name.

5 To spread a view across several screens, select **Combine equal observers and span across displays** and select the same observer for the windows/displays.

Interim result

You configured specific views for each display.

Part 4

To activate the full screen configuration

1 In **Motion Player**, select play or start a simulation on the **Home** ribbon

2 To activate the full screen mode, you have the following options:

- On the **Home** ribbon, click **Views – Single Full Screen**.
- On the **View** ribbon, click **Full Screen – Single Full Screen**.
- Press **F11**.

3 Press **Esc** to exit full screen mode.

Interim result

You activated the full screen configuration in a simulation.

Result

You configured the full screen displays and selected views and observers for each display. You also activated full screen mode during a simulation.

Related topics

HowTos

How to Configure a MotionDesk Screen for Sensor Simulation (MotionDesk Sensor Simulation Control 🕮)

References

How to Configure a Stereo Display

Objective

A stereo display provides views of the scene which can be used for a right and left eye or camera to get a 3-D view of the virtual world.

Basics

In the MotionDesk options, you can select the screens used for the stereo split screen display to show the active visualization view or a selected observer.

Note

If you selected the same screens for sensor simulation as for full screen mode, the sensor composition window is hidden behind the full screen display. You can press `Esc` to close full screen or stereo mode, or switch back to MotionDesk by pressing `Alt+Tab`.
For more information, refer to How to Configure a MotionDesk Screen for Sensor Simulation (MotionDesk Sensor Simulation Control 🕮).

Tip

In full screen or stereo mode, you can move around the 3-D View during the simulation or motion player playback. For more information, refer to Overview of the Observer Navigation Mode on page 85.

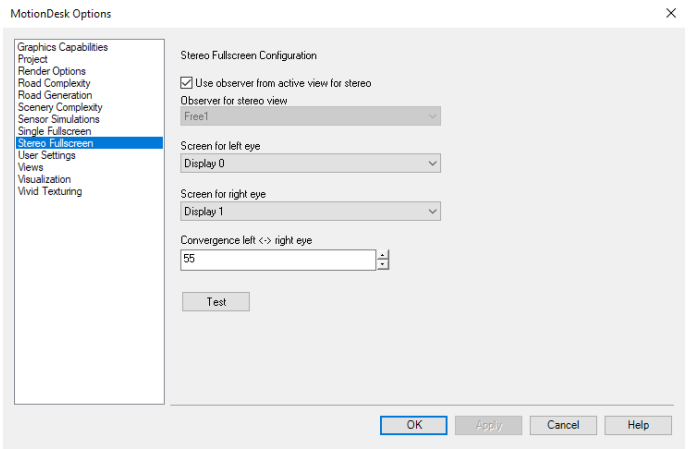
Preconditions

The MotionDesk PC must have two displays.

Method

To configure a stereo display

1 On the File ribbon, click Options - Stereo Fullscreen.



2 Select Use observer from active view for Stereo.

Tip

You can also clear Use observer from active view for Stereo and select an observer for the stereo mode in the Observer for stereo view list.

3 Select the screens for the left and right eyes.

4 Select the distance between the eyes or cameras in **Convergence left <-> right eye**

   For details of the stereo mode properties, refer to Stereo Fullscreen Options Page on page 212.

5 Click **Test**.

   The selected active visualization view in the **3-D View** is displayed in split screen stereo mode on the two selected displays. Press **Esc** to close the test windows.

6 Click **OK** to close the dialog.

7 To activate the stereo screen mode during a simulation or Motion player playback, you have the following options:
   - On the **View** ribbon, click **Full Screen – Stereo Full Screen**.
   - Press **F12**.

8 Press **Esc** to exit full screen mode.

Result

You configured the stereo screen displays and selected the observer. You also activated stereo mode during a simulation.

Related topics

HowTos

How to Configure a MotionDesk Screen for Sensor Simulation (MotionDesk Sensor Simulation Control 🕮)

References

Working with Observers

Introduction

The viewpoints in a scene are defined by observers. Observers function like cameras in the scene. Their positions and orientations can be set up in different ways to be used when navigating through the scene and in the simulation.

Where to go from here

Information in this section

Basics of Observers and the ModelDesk Observer

Introduction

MotionDesk provides a number of observers to navigate through the scene and to follow movable objects in the scene to view the simulation animation from different perspectives.

Free1 and Free2 observers

The Free1 and Free2 observers are system observers that are available when you create a new MotionDesk experiment.

You use the free observers to navigate around the 3-D View to create and analyze the scene. You can move the observer in the scene using the mouse or by specifying the x-, y-, and z- position and orientation properties. The position and orientation of the free observers remain when you select another observer and then later return to them. You cannot rename, delete, or attach motion data from the simulation to movable objects to the free observers.

For more information, refer to How to Select an Observer on page 84

ModelDesk Observer

The ModelDesk Observer is an observer whose position can be specified in ModelDesk and displayed in the MotionDesk 3-D View.

You can use the ModelDesk observer to examine a specific area in the scene when modeling a road network in ModelDesk. You must first generate the scene to display it in MotionDesk. You can move, edit, or delete the ModelDesk observer. If the observer does not exist, ModelDesk creates a new one which is displayed in the Scene Navigator.

To control the ModelDesk observer, refer to How to Examine the Road Network in MotionDesk (ModelDesk Road Creation 🕮).

Default and custom observers

You can create additional observers to help navigate around the scene and for use when displaying the simulation. The observers are added to the Scene Navigator.

You can move, edit, or delete the default and custom observers and change the movable object the observers are attached to. You can also lock the observers in position to avoid accidentally moving the observer when navigating around the scene.

- Default observers: You can create a set of MotionDeskdefault observers. The default observers include observers that follow a movable object in the scene during the simulation and fixed observers that are anchored to a point in the scene. When you create the default observers, you select the movable object in the scene to attach the observers to.
- Custom observers: You can create single custom observers. You can select to fix the observers to a position in the scene or attach them to a movable object.

Related topics

Basics

Overview of the Observer Navigation Mode....85
Scene Generation (MotionDesk Scene Creation 🕮)

HowTos

How to Examine the Road Network in MotionDesk (ModelDesk Road Creation 🕮)
How to Select an Observer....84

References

Show in MotionDesk (ModelDesk Road Creation 🕮)

Default Observers

Introduction

MotionDesk contains some default observers. You can also create your own defaults.

Default observers

You can edit the properties of default observers. The following table shows the translations and orientations of the default observers:

Observer	Observer's Translation	Observer's Orientation
Side	Follows the selected object	Views selected object from side
Front	Follows the selected object	Views selected object from front
Rear	Follows the selected object	Views selected object from behind
Rear Smooth	Follows the selected object smoothly	Views selected object from behind
Driver	Follows the selected object	Views from driver's seat
Driver Smooth	Follows the selected object smoothly	Views from driver's seat
Near	Follows the selected object	Orientation constant in the scene
Top	Follows the selected object	Views selected object from above
Overview	Follows the selected object	Views selected object from above
Fixed Overall View	Fixed	Views everything from above
X-Follow	Follows the translation in x direction of the selected object	Views selected object
Y-Follow	Follows the translation in y direction of the selected object	Views selected object
Marshall Trackside	Fixed	Views selected object

Related topics

HowTos

How to Create Default Observers..... 81

References

Observer Properties..... 200

How to Create Default Observers

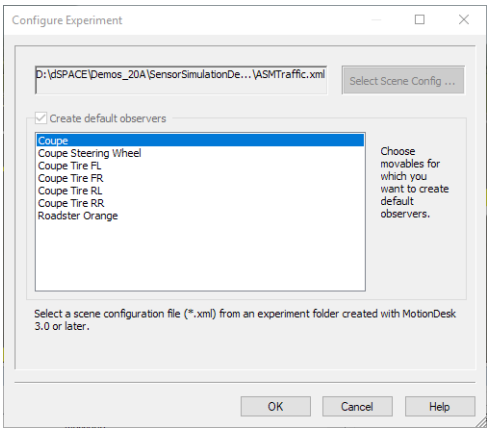
Objective

You can create MotionDesk default observers to navigate around the scene and to follow the movable objects in a simulation.

Method

To create default observers

1 On the Observation ribbon, click Observer – Create Default Observers.
MotionDesk opens the Configure Experiment dialog.



2 In the dialog, select a movable object.

3 Click OK.

Result

MotionDesk creates a set of default observers. All the new observers that can follow a movable object are assigned to the selected movable object. The '#' sign is appended to the observers' default names to give them unique names.

You can use the new observer in the scene. Double-click the observer's name in the Scene Navigator.

For descriptions of the default observers, refer to Default Observers on page 81.

Related topics

Basics

Default Observers....81

References

Create Default Observers....147

How to Create and Configure a Custom Observer

Objective

To get observers that fit your requirements, you can create new ones or reconfigure the defaults.

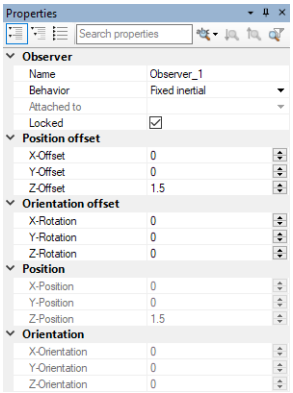
Method

To create and configure a custom observer

1 You can create a new observer from scratch or use an existing observer as a template:

- To create an observer from scratch: In the **Scene Navigator** on the **Observers** item, open the context menu and click **Create**.
- To use settings of an existing observer: In the **Scene Navigator**, select an observer and go to the **Scene** ribbon and click **Clipboard – Copy** and afterwards **Paste**.

MotionDesk creates a new observer. The **Properties** pane displays the property values of the observer.



2 In the **Name** property, enter a unique name for the observer.

3 In the **Behavior** property, select one observer type. For more information on the observer types, refer to Observer Properties on page 200.

4 If the observer is to follow a movable object, select this and then a movable object in the **Attached To** property.

5 In the other properties, select or enter values for the offset position and orientation of the observer. When values are changed, the **3-D View** is updated immediately.

Result

You can use the new observer in the scene. Double-click the observer's name in the **Scene Navigator**.

Related topics

References

Create (Observer) 145
Observer Properties 200

How to Select an Observer

Objective

You can select one observer for each window of the 3-D View to observe a scene from different positions.

Selecting and using an observer

In the Scene Navigator, you can double-click any of the observer views to display the scene in the 3-D View from a different perspective. For example, you can view the scene from the view of the driver of a vehicle or from behind or above the vehicle. When you start the simulation, the observer moves with the animation from the selected observer view position.

For more information on observers, refer to Basics of Observers and the ModelDesk Observer on page 79.

Method

To select an observer

1. Click the 3-D View window where you want to select an observer.
2. Open the context menu in the window and select an observer.

   Or

   Double-click the observer's name in the Scene Navigator.

   Or

   On the Observation ribbon, click Observer – Active and select the observer.

Result

The selected observer is used in the window.

Related topics

Basics

Basics of Observers and the ModelDesk Observer 79
Default Observers 81

References

Active Observer 138

Navigating Through the Scene with Observers

Introduction

You can navigate through the scene in the 3-D View with the observers using the mouse or by setting the properties for the observers.

Where to go from here

Information in this section

Overview of the Observer Navigation Mode

Introduction

You can use the observer views in MotionDesk to move around the 3-D View with the observer navigation mode.

Zone and observer navigation

MotionDesk provides two modes to navigate around the scene. You can select observer and zone navigation in the **Observer** ribbon.

The observer navigation allows you to move and look around the scene with the mouse similar to being inside the scene. You can move around the scene entirely

with the mouse and the three mouse buttons. A turbo function allows you to move quickly around larger scenes. You can also easily rotate around an object in the scene in front of the observer.

For more information on the zone navigation mode, refer to Working with Observers in Zone Navigation Mode.

Overview of the observer navigation mode

You can use the mouse to navigate freely around the 3-D View in MotionDesk.

You can select one of the free observers and move the observer along each of the x-, y -and z- axes. You can move upwards and downwards above the scene and forwards, backwards, left, and right through the scene using the mouse buttons and the mouse wheel. You can also use the W, A, S, D keys on the keyboard to move around the scene.

You can also lock and unlock the position and orientation of the observers. When locked, the observer view returns to the original position when the mouse button is released after moving around the 3-D View.

You can also rotate the observer around an object.

A center point and each of the axes are displayed in the scene when moving around an object in the scene in this mode.

Tip

If you use large scenes, you can fly to an object and move the observer from this new position.
Select an object in the Scene Navigator and select Fly to Object in the mouse context menu or press `Ctrl F`.
You can also press `SHIFT` on the keyboard to move faster through the scene using the turbo mode when moving around the scene with the mouse or keyboard.

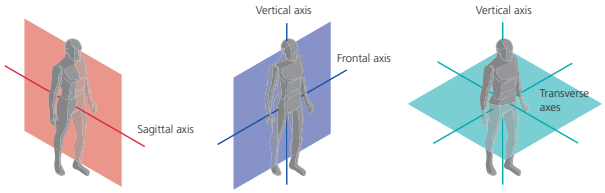
The observer navigation mode can also be used to move around the 3-D View in the single and full screen display modes.

Illustration

The planes of human movement used by the observer navigation are shown in the following diagram.

Planes of movement

- Sagittal plane: Move forwards and backwards.
- Frontal plane : Move up, down, left & right.
- Transverse plane: Rotation on the longitudinal (vertical) axis along the transverse plane.



Sagittal plane (Forward and backwards movement)
This is the head to toe plane from front to back

Frontal plane (Up, down, left & right movement)
This is the head to toe plane from left to right

Transverse plane (Rotation on the longitudinal (vertical) axis)
This is the square plane at waist level

Selecting the observer navigation mode

The observer navigation mode is enabled when you upgrade MotionDesk. You can switch between the observer navigation mode and the zone navigation mode. You can also configure the movement speed of the observer navigation mode. Refer to How to Select the Observer Navigation Mode and Speed on page 87.

Related topics

Basics

Basics of Observers and the ModelDesk Observer.....79
Editing a Scene in the 3-D View (MotionDesk Scene Creation 🕮)
Working with Observers in Zone Navigation Mode.....96

References

Fly to Object (MotionDesk Scene Creation 🕮)

How to Select the Observer Navigation Mode and Speed

Introduction

You can select the observer navigation mode and configure the speed for moving around the 3-D Viewne.

Overview

In the MotionDesk **Observation** ribbon, you can switch between the observer navigation mode and the zone navigation mode.

You can also set the speed for moving around the scene in the observer navigation mode. This controls how fast the observer moves through the scene when you navigate with the mouse and keyboard.

- To select the navigation mode, refer to Method 1 on page 88.
- To set the movement speed, refer to Method 2 on page 88.

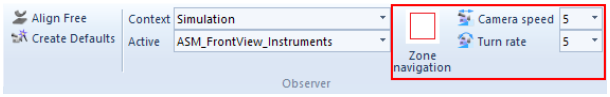
Method 1

To select the navigation mode

1 Open a MotionDesk project and select the Observation ribbon.

2 Select Observer - Zone navigation to switch between the standard zone navigation and the observer navigation modes.

When the observer navigation mode is active, the Zone navigation button is not selected. The Camera speed and Turn rate list fields are enabled.



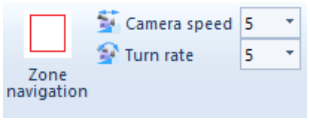
Result

You switched between the standard zone navigation mode and the observer navigation modes.

Method 2

To configure the observer movement speed

1 Open a MotionDesk project and select the Observation ribbon.



2 Select Observer - Camera speed to adjust the speed at which you move through the scene.

3 Select Observer - Turn rate to adjust the rotation speed of the observer on a fixed position to look around the scene.

4 Use the mouse to move around the 3-D View to check the movement speed is suitable.

Note

The Camera speed and Turn rate settings apply to all observer views. The speed and turn rate can be changed only in the observer navigation mode.

Result

You changed the observer movement speed.

Whats next

You will use the observer navigation mode to move around the MotionDesk 3-D View using the mouse. Refer to How to Move the Observer Using the Mouse.

Related topics

Basics

Overview of the Observer Navigation Mode....85

HowTos

How to Move the Observer Using the Mouse....89

References

Camera Speed (Observer)....143
Turn rate (Observer)....168
Zone and Observer navigation....170

How to Move the Observer Using the Mouse

Introduction

You can select an observer to move freely around the 3-D View with the mouse, mouse buttons, and mouse wheel using the observer navigation mode.

Overview

You can select one of the free observers and move along each of the x-, y- and z-axes of the scene using the mouse and mouse buttons. You can also move quickly through large scenes and rotate the observer view for a better view of an object in the scene.

- To move forward and backward through the scene along the transverse plane and look left and right, refer to Method 1 on page 90.
- To move upward and downward above the scene and sideward across the scene, refer to Method 2 on page 90.
- To rotate the observer head motions to look left right, up, and down from a specific point in the scene, refer to Method 3 on page 91.
- To move quickly through large scenes, refer to Method 4 on page 91

Note

The cursor arrows disappears when you press a mouse button. You cannot move the cursor outside the 3-D View.

Default and custom observers In the observer navigation and the zone navigation modes, you can also move the position and orientation of the default and custom observers that are fixed or attached to motion data.

By default, the default and custom observers are locked to a position or to the offset from the motion data. This avoids accidental movement of the observer when moving around the scene.The view returns to the original position when you release the mouse button.

To change the position and orientation of the observer permanently, you must first clear the Locked checkbox for the observer in the properties dialog.

You can also manually change the observer position in the properties pane, for example, you can change the offset and rotation from the initial position determined by the attached data stream. If the observer behavior is Free, you can specify the position in meters and the orientation in degrees based on the scene and the horizon.

Note

You cannot edit the Z Rotation value of the Follow rotation behavior in the observer properties. You can edit the position offset and the x- and y-rotation properties.

Method 1

To move forward, and backward through the scene

1. Select the Free1 or Free2 free observer in the Scene Navigator.
2. Click on the 3-D View.
3. Click and hold the left mouse button and move the mouse forward.
   The observer view moves directly forward through the scene along the transverse plane.
4. Click and hold the left mouse button and move the mouse backward.
   The observer view moves directly backward through the scene along the transverse plane.

   Note

   The observer does not move upward and downward above the scene. The height above the scene is maintained.

5. Click and hold the left mouse button and move the mouse left and right.
   The observer rotates left and right on the fixed vertical axis to display a view of the scene to the left and right of the object.

Result

You moved the observer forward, and backward through the MotionDesk 3-D View. You also rotated the observer left and right on the fixed vertical axis.

Method 2

To move upward, downward and sideways in the scene

1. Select the Free1 or Free2 free observer in the Scene Navigator.
2. Click on the 3-D View.
3. Click and hold the middle mouse button or mouse wheel and move the mouse forward.
   The observer view moves directly upward, rising above the scene.

4 Click and hold the middle mouse button or mouse wheel and move the mouse backward.

The observer view moves directly downward, toward the ground of the scene.

5 Click and hold the middle mouse button or mouse wheel and move the mouse left and right.

The observer moves left and right in a side step motion across the screen.

6 Roll the mouse wheel forward and backward.

The observer view moves forward and backward in a zoom in and out motion in the direction the observer faces.

Result

You moved the observer upward, downward and sideways in the MotionDesk 3-D View. You also zoomed in and out.

Method 3

To rotate the observer view

1 Select the free observer Free1 or Free2 in the Scene Navigator.

2 Click on the 3-D View.

3 Click and hold the right mouse button and move the mouse forward.

The observer rotates on the frontal axis to look upward toward the sky, similar to rolling the viewers head backward.

4 Click and hold the right mouse button and move the mouse backward.

The observer rotates on the frontal axis to look downward toward the ground, similar to rolling the viewers head forward.

Note

The viewer remains in a fixed position in the scene.
The observer does not move upward and downward above the scene, side step across the scene, or move forward and backward through the scene.

5 Click and hold the right mouse button and move the mouse left and right.

The observer view rotates left and right on the vertical axis, similar to turning the viewers head left and right.

Result

You rotated the observers head in the MotionDesk 3-D View to look left, right, up, and down

Method 4

To move through the scene using turbo mode

1 Select the Free1 or Free2 free observer in the Scene Navigator.

2 Click on the 3-D View.

3 Press and hold the **Shift** keyboard key and click the **left** mouse button. Move the mouse forward and backward.

   The observer view moves directly forward and backward through the scene at high speed.

4 Press and hold the **Shift** keyboard key and click the **middle** mouse button. Move the mouse forward and backward.

   The observer view moves directly upward and downward in the scene at high speed.

5 Press and hold the **Shift** keyboard key and click the **middle** mouse button. Move the mouse left and right.

   The observer view side steps left and right across the scene at high speed.

Note

The high speed turbo mode does not affect the rotation of the observer, for example, when clicking the right mouse button.

Result

You used the high-speed mode to move quickly through the MotionDesk **3-D View**.

Whats next

You will use the new observer navigation mode to move around the MotionDesk **3-D View** using the keyboard. Refer to How to Move the Observer Using the Keyboard.

Related topics

Basics

Overview of the Observer Navigation Mode 85
Working with Observers in Zone Navigation Mode 96

HowTos

How to Move the Observer Using the Keyboard 93

References

Observer Properties 200

How to Move the Observer Using the Keyboard

Introduction

You can select an observer to move freely around the 3-D View with the keyboard using the observer navigation mode.

Note

By default, the default and custom observers are locked to a position or to the offset from the motion data. This avoids accidental movement of the observer when moving around the scene. The view returns to the original position when you release the mouse button.
To change the position and orientation of the observer permanently, you must first clear the Locked checkbox for the observer in the properties dialog.

Method

To move the observer using the keyboard

1. Select the free observer Free1 or Free2 in the Scene Navigator.
2. Click on the 3-D View.
3. Select and hold any mouse button and press and hold the W keyboard key.
   The observer view moves directly forward through the scene in the direction that the observer faces.

   Note

   If the observer does not face in a level direction but looks up or down, the observer also moves upward or downward in addition to forward and backward. The height above the scene is not maintained.

4. Select and hold any mouse button and press and hold the S keyboard key.
   The observer view moves backward through the scene.
5. Select and hold any mouse button and press and hold the A keyboard key.
   The observer moves left in a side step motion across the screen.
6. Select and hold any mouse button and press and hold the D keyboard key.
   The observer moves right in a side step motion across the screen.

   Tip

   You can combine the keyboard keys and the mouse to move in multiple directions in the scene and have a better view of an object.
   For example, press the right mouse button and move the mouse left and right to look left and right. Then press W and S to move toward and away from the object.

Result

You moved the observer forward, backward, and sideways in the MotionDesk 3-D View.

Whats next

You will rotate around objects in the scene. Refer to How to Rotate Around an Object in Front of the Observer (Trackball) on page 94.

Related topics

Basics

Overview of the Observer Navigation Mode....85
Working with Observers in Zone Navigation Mode....96

References

Observer Properties....200

How to Rotate Around an Object in Front of the Observer (Trackball)

Introduction

You can rotate around an object in the 3-D View around a point in front of the observer.

Overview

You can rotate around an object in the 3-D View. You move the observer around a point in the scene in front of the observer.

This can be used to move around an object in the scene, for example, a vehicle. The observer moves in a circle around the object.

You can rotate along two axes:

- Vertical axis: You move left and right around the object, for example, as if the observer walks around a vehicle.
- Frontal axis: You move up and down around the object, for example, as if the observer flies over a vehicle from front to back and underneath the vehicle to return to the starting point.

You activate this rotation by pressing the Ctrl keyboard button and the left mouse button.

You can also use the mouse wheel to move forward and backward through the scene, for example to move closer to the object around which you want to rotate using the Ctrl key.

Note

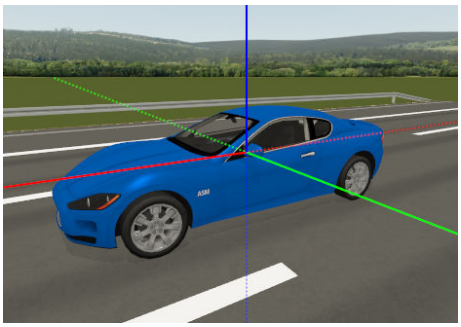
When you press Ctrl, the x-, y-, and z- rotation axes are shown in the scene. The axes meet at the center point of the object around which you rotate.

Method

To rotate around an object in front of the observer

1. Select the Free1 or Free2 free observer in the Scene Navigator.
2. In the Scene Navigator, select an object, for example, a vehicle and select Fly to object in the mouse context menu to move to the vehicle in the 3-D View.
3. Select and hold the Ctrl keyboard button and the left mouse button. Move the mouse left across the scene.

   The axis is displayed in the scene at the center of the object and the observer moves left in a circle around the object.



4. Repeat the previous step and move the mouse right.

   The axis is displayed in the scene and the observer moves right in a circle around the object.
5. Select and hold the Ctrl keyboard button and the left mouse button. Move the mouse upward.

   The axis is displayed in the scene and the observer moves up in an arc around the object.
6. Repeat the previous step and move the mouse downward.

   The axis is displayed in the scene and the observer moves down in an arc around the object.

Note

You can move the observer to change the center position around which you rotate. For more information, refer to How to Move the Observer Using the Mouse on page 89.

Result

You rotated around an object in front of the observer in the 3-D View.

Whats next

You will learn to work with zone navigation in the 3-D View. Refer to Working with Observers in Zone Navigation Mode on page 96.

Related topics

Basics

Overview of the Observer Navigation Mode 85

HowTos

How to Move the Observer Using the Mouse 89

References

Fly to Object (MotionDesk Scene Creation 🕮)
Observer Properties 200

Working with Observers in Zone Navigation Mode

Introduction

You can use the zone navigation mode to freely navigate through the scene in the 3-D View by using a free observer and unlocked default or custom observers.

Zone and observer navigation

MotionDesk provides two modes to navigate around the scene. You can select observer and zone navigation in the Observer ribbon.

The zone navigation mode allows you to move around the 3-D View with the mouse and keyboard. You use the keyboard keys to move the mouse in different directions. You can specify the movement and turn rates of the observers in the Visualization Options Page on page 214.

Note

The zone navigation mode is to be removed in an upcoming release.

For more information on the observer navigation mode, refer to Overview of the Observer Navigation Mode.

Controlling a free or unlocked default or custom observer

A free observer can be placed anywhere in the scene. Default and custom observers can be fixed in the scene or attached to motion data. When unlocked, default and custom observers can also be moved or rotated using the zone navigation mode.

You can use the following instructions to move the observers in zone navigation mode.

Note

You must first activate Zone navigation in the Observation ribbon. By default Observer navigation is activated.
For more information, refer to How to Select the Observer Navigation Mode and Speed.

Observer Action	User Action
Moving an observer forward or backward	Drag the mouse pointer forward or backward in the 3-D View.
Turning an observer sideways	Drag the mouse pointer sideways in the 3-D View.
Moving an observer upwards or downwards	Hold down the **Shift** key and drag the mouse pointer forward or backward in the 3-D View.
Moving an observer sideways	Hold down the **Shift** key and drag the mouse pointer sideways in the 3-D View.
Changing an observer's pitch	Hold down the **Tab** key and drag the mouse pointer forward or backward in the 3-D View.
Rolling an observer	Hold down the **Tab** key and drag the mouse pointer sideways in the 3-D View.
Aligning an observer	On the menu bar, select Visualization - Observers - Align Observer.
Moving an observer to a 3-D object	Double-click a 3-D object in the Scene Navigator.

Tip

You can specify the minimum and maximum rate for moving and turning on the Visualization Options Page on page 214 page.

Dead zone There is a dead zone around the free observer to make it easier to handle. The observer moves only if the mouse pointer is outside the dead zone. The 3-D View visualizes the dead zone as a rectangle which appears when you drag the mouse pointer.

Tip

Fly to Object
You can right-click any object in the Scene Navigator and select Fly to Object to move the free observer to the selected 3-D object. For example, when you add small objects to the scene you can move directly to them using this command.

Properties of free observers

When you select a free observer, the Properties pane is displayed. This shows the position and orientation of the observer. You can modify the properties to move the free observer to a new position.

Related topics

Basics

Basics of Observers and the ModelDesk Observer 79
Overview of the Observer Navigation Mode 85

HowTos

How to Select the Observer Navigation Mode and Speed 87

References

Align Free Observer 141
Fly to Object (MotionDesk Scene Creation 🕮)
Visualization Options Page 214

Storing Motion Data

Introduction

Motion data arriving from the simulation (real-time or PC-based Simulink simulation) is written to a frame buffer. The contents of the frame buffer can be written to a file. This file can be used to replay the animation.

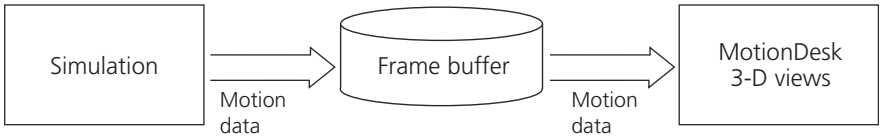
Where to go from here

Information in this section

How to Store Motion Data

Frame buffer

Motion data arriving from the simulation (real-time or PC-based Simulink simulation) is written to a frame buffer, see the following illustration.



The size of the buffer is defined when you select the data source. Refer to How to Select the Data Source on page 16. The frame buffer is a ring buffer. If it is full, new data overwrites the old data starting at the beginning of the buffer. This guarantees that once the buffer is filled, you can use the data of the entire buffer.

Method

To create and store motion data

1. On the **Home** ribbon, click **Simulation – Go Online** to start the visualization.
2. On the **Motion Player**, watch the **Frame**. When the values recur, the frame buffer is filled.
3. On the **Home** ribbon, click **Simulation – Go Offline**.
4. On the **Home** ribbon, click **Save As – Store**.
   A standard **Save As** dialog opens.

5 Select a path and specify a file name for the MDF file.

Note

Do not use Asian characters, diacritic marks (such as ö or ó), and special characters (such as & : < > ' ") in the file path. MotionDesk does not support these characters.

Result

MotionDesk stores the frames from the buffer to an MDF file.

Next step

You can use the saved MDF file to replay an animation. Refer to How to Replay an Animation on page 103.

Related topics

Basics

Structure of the Motion Data File 100

References

Go Offline 183
Go Online 182
MDF 183

Structure of the Motion Data File

Basics

The motion data file (MDF file) consists of a header block at the beginning, a comment block and one or more frame blocks, depending on the number of time steps. It must have the format described below.

Header block

The first line of the motion data file consists of the name of the product, its version and the current year. This line is part of the header block for the number of movable objects of the animation. The following table shows the keywords that are used between the `BEGINHEADER` and `ENDHEADER` keywords to hold information on the movable objects and signals.

Keyword	Description
`NPARTS`	Number of movable objects and signals
`NBODIES`	Number of movable objects
`BODY<n>`	Name of the <n>. body specified in the real-time model
`NSIGNALS`	Number of signals

Keyword	Description
SIGNALNAME<n>	Name of the signal <n> specified in the real-time model
SIGNALUNIT<n>	Unit of the signal <n> specified in the real-time model

Comment block

A block of comments can be added at the beginning of the motion data file after the header block and between each frame. The comment lines must be introduced by "C", "c", or "!".

Frame block

The frame block is introduced by a frame header, a single-lined text which contains the **Time** keyword followed by the value of the time step, for example, **Time: 0.2000 s**. This allows MotionDesk to display the MDF frames in real time.

Tip

MDF files that were created before MotionDesk 2.0 do not contain time stamps. MotionDesk therefore ignores the time step values for these MDF frames and displays them as fast as possible.

The rest of the frame block contains an animation transformation for all element movements. Each animation transformation consists of a 3×3 rotation matrix and a 3×1 translation vector relating to the inertial system or to the world coordinate system.

The common representation of a homogeneous 4×4 transformation matrix is:

```
rot(1,1)  rot(1,2)  rot(1,3)  tran(1)
rot(2,1)  rot(2,2)  rot(2,3)  tran(2)
rot(3,1)  rot(3,2)  rot(3,3)  tran(3)
0         0         0         1
```

As the information in the last line of the 4×4 transformation matrix is constant, only the first three lines of the matrix are stored in the MDF file.

All the motion data of each frame (one 3×1 translation vector and one 3×3 rotation matrix for each motion) make a total of 12 floating-point values per motion. These coordinates must be saved contiguously in the ASCII format in the following order:

```
tran(1)  tran(2)  tran(3)  rot(1,1)  rot(1,2)  rot(1,3)
rot(2,1)  rot(2,2)  rot(2,3)  rot(3,1)  rot(3,2)  rot(3,3)
```

Example

The following example shows an MDF file:

Header block: Holds information on the movable objects and signals contained in the file. The header block has to start in the first line and the first column of the file.

```
MotionDesk 3.0 2011
TRACKSAFE
BEGINHEADER
CFRAMES = 500
NBODIES = 3
BODY1 = MyFirstBody
BODY2 = MySecondBody
BODY3 = MyThirdBody
NSIGNALS = 2
SIGNALNAME1 = MyFirstSignal
SIGNALNAME2 = MySecondSignal
SIGNALUNIT1 = m
SIGNALUNIT2 = s
ENDHEADER
```

Comment block as file header. The comment lines must be introduced by "C", "c", or "!".

```
C data record from 1.6.2011
C car simulation ACSL
C step size of the integration 0.010 s
```

1st frame block: frame heading.
Translation vector and rotation matrix of all movable objects at the time of 0.02 s.

```
Time: 0.020000 s
0.000 0.000 0.000 1.000 0.000 0.000
0.000 1.000 0.000 0.000 0.000 1.000
...
```

2nd frame block: frame heading.
Translation vector, rotation matrix, etc.

```
Time: 0.090000 s
0.000 0.000 0.000 1.000 0.000 0.000
0.000 1.000 0.000 0.000 0.000 1.000
...
```

Postprocessing an Animation

Introduction

An animation can be replayed after a simulation. This requires a motion data file that can be replayed and used to generate a video.

Where to go from here

Information in this section

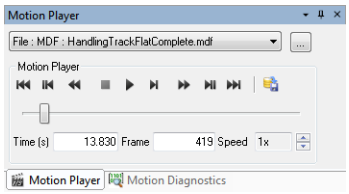
How to Replay an Animation

Objective

If you have saved the motion data of an animation to a file, you can replay the animation offline. This is specially useful for analyzing motion data frame by frame after simulation.

Basics

You can control the replay with the Motion Player, see the following illustration.



Preconditions

Before you can replay an animation, you have to prepare the scene and create a motion data file. For more information, refer to How to Store Motion Data on page 99. MotionDesk can also replay an MDF file generated by RealMotion.

Restrictions

- For MDF files generated before MotionDesk 2.0, no time stamps for frames are recorded in the motion data file. The animation runs as fast as possible, so it may run faster than the simulation.
- You can only replay track-safe animations in slow or fast motion. Refer to Basics of the Multi-Track Mode on page 109.

Part 1

To start a replay

1 Click (Start Replay) to start the animation for a single loop through the frame buffer.

Click (Start Replay (Continuous Loop)) to start the animation for an endless loop.

Part 2

To stop a replay

1 Click (Stop).

Part 3

To view frames step-by-step

1 Click (Next Frame) to view the next frame.

Click and hold (Next Frames) to view the next few frames.

Click (Previous Frame) to view the previous frame.

Click and hold (Previous Frames) to view the previous few frames.

Click (First Frame) to view the first frame.

Click (Last Frame) to view the last frame.

Part 4

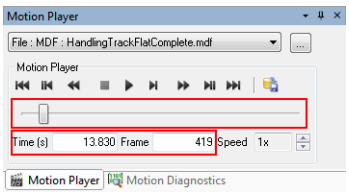
To move to a specific time or frame in the animation

1 Stop the animation.

You can move the slider and adjust the time and frame only when the Motion Player is stopped. The slider and the fields are disabled when the player is running the animation.

2 To move through the animation, move the **Slider** forward and backward to a specific **Time** and **Frame**.

The scene moves forward and backward through the animation until you release the mouse button. The time and frame values also change when you move the slider.



3 To jump to a new point in time in the animation, in the **Time** field, enter the time in seconds and press the enter key.

The scene displays the stopped animation at that specific point in time. The frame number is also changed.

4 To jump to a new frame in the animation, in the **Frame** field, enter the frame number and press the enter key.

The scene displays the stopped animation at that specific frame. The time in seconds is also changed.

5 Start the simulation to start the replay at the point you selected using the **Slider**, **Time**, or **Frame** number.

Part 5

To replay an animation in slow or fast motion

1 In the **Speed** box, select the speed of the animation replay.

You can select 2× up to 32× for fast motion and 1/2× down to 1/32× for slow motion.

Result

MotionDesk replays the animation in the selected way.

Related topics

References

Motion Player 184

Generating a Video of an Animation

Introduction

An animation can be replayed only on computers which have MotionDesk installed. If you want to show the animation on other computers, you have to generate a video.

Basics of video

Frame duration The frame duration is the time for which one frame of a video is displayed on screen. In a motion data file, the time is not constant, because each frame is displayed as fast as possible. The frame duration must be

estimated. You can measure the whole time for an animation and divide it by the number of frames.

Frame rate The frame rate is the number of frames per second which are displayed in the video. Normally, this is 25.

Enable logo You can also enable a logo that is displayed on the saved video. To add a new logo, save the picture file in PNG format in `<MotionDesk_InstallationPath>\MotionDesk\Assets\Pictures\VideoLogo.png`.

Codec format Codec format is the file format for videos. MotionDesk provides all the codec formats which are installed on your PC. Use a format that is also available on the computer that you want to use for replaying.

As of MotionDesk version 3.9 dSPACE recommends the Xvid MPEG-4 Codec.

The file format of the generated video is Audio Video Interleaved (AVI) or Moving Picture Experts Group 4 (MPEG-4).

Tip

Do not use the uncompressed format. The size of the generated file is much larger than the size of a file generated in compressed codec format.

Resolution When generating the video, MotionDesk replays the animation and captures the 3-D View. The size of the 3-D View is equivalent to the resolution of the video. The maximum resolution depends on the maximum size of the 3-D View.

Tip

If you want to generate a video with a high resolution, you can hide the Navigator, tool window, and Motion Player. Keep in mind that generating a really high resolution video, for example, 1024 × 786, needs a lot of calculation power, and the generated video might be replayed jerkily.

Setting the video properties

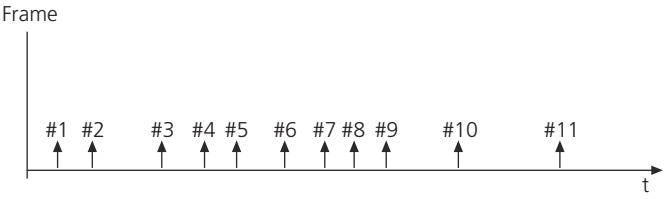
When generating a video, you have to set its export properties. Whether you have to set the frame duration and frame rate depends on the MotionDesk version used to record the MDF files or replay the animation.

MotionDesk 2.0 and higher MotionDesk automatically selects the frame duration and the frame rate. You do not need to change these values.

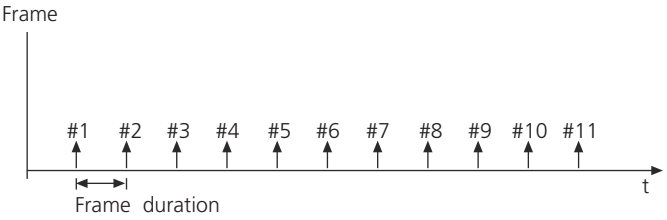
MotionDesk versions earlier than MotionDesk 2.0 Time stamps for frames were not generated in real time. You have to define the frame duration and the frame rate. This applies to MDF files, Simulink simulations and multi-PC

real-time applications. Real-time applications running on the DS1006, DS1007, or SCALEXIO systems support time stamps.

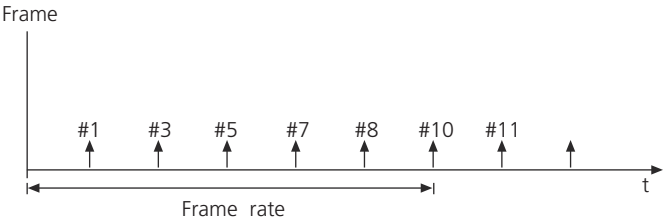
The following illustrations show the procedure. The first illustration shows the frames of a motion data file. The time between the frames is not constant. It depends on the number of objects that are displayed.



The second illustration shows the same frames, but the time between them is constant (= frame duration).



The third illustration shows the frames of the video file. Frames are duplicated or deleted to get the specified frame rate. In this example a frame rate of only 6 fps is specified, so some frames are deleted.



Related topics

HowTos

How to Generate a Video 107

How to Generate a Video

Objective

If you want to show an animation on a computer without MotionDesk, you can generate a video.

Preconditions

Before you can generate the video, you have to animate the scene to fill the frame buffer.

Method

To generate a video

1. Replay the animation.
2. If the 3-D View is split, click the 3-D view which you want to use for video generation.

   After selection the 3-D view is framed in red.
3. On the Home ribbon, click Save As – Export Video to open the Video Export Properties dialog.
4. Specify the properties.

   If you use a MotionDesk version earlier than MotionDesk 2.0, you have to define the frame rate and frame duration. If you work with MotionDesk 2.0 and higher, MotionDesk automatically selects these values.

   You can also enable a logo that is displayed on the saved video.
5. To change the logo, save a PNG picture file in `<MotionDesk_InstallationPath>\MotionDesk\Assets\Pictures\VideoLogo.png`.
6. Select a supported codec. As of MotionDesk version 3.9, dSPACE recommends the Xvid MPEG-4 Codec.
7. Select the file name, location, and extension for the video.
8. Click Generate to start the video generation.

Result

MotionDesk sets the window size of the view to the values defined by the properties and starts the animation. Each frame is saved to the specified file in the time steps defined by the properties. The video is generated and is stored in the selected file location.

Related topics

Basics

Generating a Video of an Animation 105

HowTos

How to Replay an Animation 103

References

Video (Export) 179

Comparing a Simulation With Recorded Tracks

Introduction

If you want to compare, for example, vehicle behavior when braking with and without ESP, you have to compare a simulation with reference tracks. Use the multi-track mode for synchronous, real-time animation.

Where to go from here

Information in this section

Basics of the Multi-Track Mode

Introduction

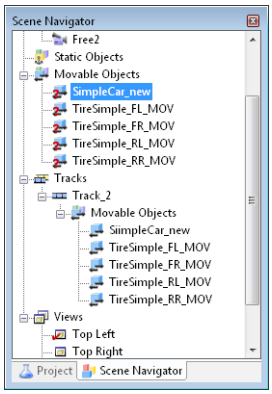
The multi-track mode allows you to compare a simulation with tracks synchronously and in real time.

Tracks

Tracks are the recorded data of simulations. You can use them as reference tracks to compare them with a synchronously running simulation.

Tracks have their own objects and observers which are cloned from the original objects and observers. The observers belonging to a track are attached to the geometries of its objects.

Cloning movable objects The movable objects of the tracks are cloned from the original movable objects. The clones are independent of the original and contain copied geometries and data connections.



All movable objects are cloned when a track is created by default. You can disable this feature and clone single movable objects manually by drag & drop, for example, if the experiment contains movable objects for two cars and you want clones of the movable objects of only one car.

Playing tracks in a loop If you play a track, its animation will stop after all the frames have been played once. If you want the track to run continuously, play it in an endless loop. You have to select this feature when you create the track.

Time stamps

Data sources contain motion data and the time stamps of the corresponding motion data frames. With time stamps, you can animate data sources in real time, which is necessary for working with the multi-track mode.

Multi-track mode

With the multi-track mode, you can replay real-time animations of data sources as reference tracks for a simulation.

You can compare up to 30 tracks with one simulation synchronously. Each track adds objects to the experiment, and they all have to be drawn. The larger the number of objects, the lower the frame rate and the animation performance.

Tip

You can hide objects to improve the animation performance. Since you have to hide each object separately and manually, add only as many tracks as you actually need.

Time offset If the simulation and the track use data sources that are identical in parts, the geometries of their movable objects are identical. You can define a time offset to distinguish them. This starts the track with a delay.

Coloring To distinguish movable objects of different tracks, you can assign different colors to the objects.

Track-safety

If you want to use a data source as a reference track for a simulation, the data source has to be track-safe. Data is track-safe if its acquisition is lossless.

Lossless data acquisition Before MotionDesk 2.0, data was not acquired at regular intervals but as soon as the previous frame was buffered and drawn. This caused irregular time intervals and missing frames.

Since MotionDesk 2.0, lossless data acquisition ensures that data is acquired completely and at regular intervals, and therefore is track-safe. This allows a smooth comparison of the simulation with the reference tracks. For details on setting lossless data acquisition, refer to General Page (MD_Communication) (MotionDesk Calculating and Streaming Motion Data 🕮).

You may get jitter if the sample time of the simulation differs from that of the track. You can avoid this by putting the MotionDesk Blockset or the Model and Sensor Interface Blockset blocks of your Simulink model into triggered subsystems. Use the same sample time for every model.

Tip

Use a sample time of 10 ms for man-in-the loop and 20 ms for automated driving maneuvers. Set the sample time to at least 10 ms to ensure that lossless data acquisition works properly.

Refer to How to Define a Sampling Rate for the Animation (MotionDesk Calculating and Streaming Motion Data 🕮).

Track synchronization

To ensure a synchronous comparison of simulation and reference tracks, MotionDesk offers track synchronization when the multi-track mode is used. Track synchronization requires lossless data acquisition and recording synchronization.

Recording synchronization If the start times of the simulation and the reference tracks differ, their movable objects may move in different areas of the scene and cannot be compared directly.

Recording synchronization ensures the synchronization of the simulation and the reference tracks. When MotionDesk is running and you reset the simulation in ControlDesk, MotionDesk clears the simulation buffer and restarts the tracks simultaneously with the simulation. This works only if all tracks were recorded with recording synchronization beforehand and recording synchronization is set for the simulation.

For details on setting recording synchronization, refer to MD_Communication (MotionDesk Calculating and Streaming Motion Data 🕮).

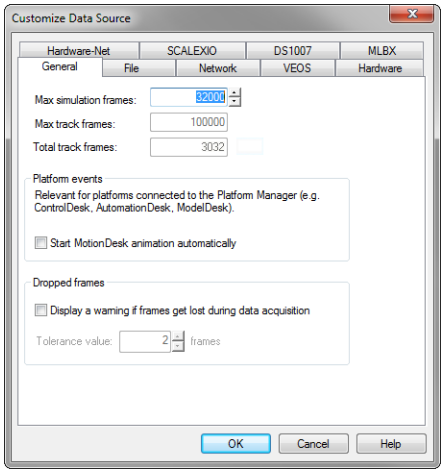
Buffer size

The maximum number of frames is spread across one simulation buffer and up to 30 track buffers.

Tip

You can check how many frames you already used. Refer to Customize Data Source on page 172.

Each track should have a buffer with at least 1,000 frames. The animation then runs long enough for a proper comparison with the simulation. Therefore, you cannot specify a simulation buffer with more than 131,000 frames.



The buffer size corresponds to a certain recording time. For example, 10 ms and 20 ms are typical sample times of a Simulink simulation. With these sample times, 1000 frames result in a recording time of 10 s or 20 s, respectively. See the table below for details:

Buffer Size [Frames]	Recording Time	
	With 10 ms Interval	With 20 ms Interval
1,000	10 s	20 s
10,000	100 s	200 s
20,000	200 s	400 s
32,000	320 s	640 s

You get the recording time in seconds by multiplying the buffer size by the interval time in seconds.

For information on setting the buffer size, refer to Customize Data Source on page 172.

Tip

If you need more than the given 131,000 frames for your tracks, contact dSPACE Support.

Related topics

Basics

Data Sources for the Multi-Track Mode..... 114

Data Sources for the Multi-Track Mode

Introduction

You can use MDF files, Simulink simulations, or real-time applications as data sources for tracks.

Simulink simulations and real-time applications have to run before you can use their simulation buffer as a data source.

MDF files

MDF files offer the following levels of track suitability:

Level	File Header	Description	MotionDesk Version	Track Safety	Recording Synchronization

1	`MotionDesk 2.0 2004`
	`BEGINHEADER`
	`TRACKSYNCHRONIZED`
	`NPARTS=14`
	`CFRAMES=2254`
	`ENDHEADER`
		The file is track-synchronized.	2.0 or later	✓	✓

2	`MotionDesk 2.0 2004`
	`BEGINHEADER`
	`TRACKSAFE`
	`NPARTS=14`
	`CFRAMES=2254`
	`ENDHEADER`
		You can use this file but will get a warning message when using it.	2.0 or later	✓	–

3	`MotionDesk 2.0 2004`
	`BEGINHEADER`
	`NPARTS=14`
	`CFRAMES=2254`
	`ENDHEADER`
		If you try to use such an MDF file as a track data source, MotionDesk will generate a warning and the assigned objects will not move.	2.0 or later	–	–

4	`MotionDesk 1.4 2004`
	`BEGINHEADER`
	`NPARTS=14`
	`CFRAMES=2254`
	`ENDHEADER`
		If you try to use such a file as a data source, MotionDesk will generate a warning and the assigned objects will not move.	Earlier than 2.0	–	–

Simulink simulations

You need the MotionDesk Blockset that comes with MotionDesk 2.0 or later to use Simulink simulations as data sources for the multi-track mode.

Limiting the frame rate You must select frame rate limiting on the Simulink page in the MD_Communication block if you want to use the multi-track mode with Simulink. The MD_Communication block of the MotionDesk Blockset transmits motion data packages to MotionDesk via the UDP/IP protocol. One motion data package equals one frame. Transmission is restricted to 50 frames per second (fps) to ensure problem-free transfer via the network. If MD_Communication is called too early, it sets the Simulink simulation to sleep mode. This stops the entire simulation for a short period of time and no frames are transmitted. This ensures that the block transmits all the data produced.

Tip

If the MotionDesk blocks are integrated in a subsystem triggered every 20 ms, MD_Communication transmits the data almost in real time.

If you have a large model such that the MD_Communication block is called less than 50 times per second, rate adjustment will not affect the simulation. However, small models may run much more slowly with rate adjustment.

MotionDesk will know from its network protocol if the Simulink simulation is track safe or not.

Note

Limiting the frame rate affects only Simulink simulations and not real-time applications.

Real-time applications

You can use real-time applications as data sources for the multi-track mode.

Multi-PC Solution You can use multi-PC real-time applications from MotionDesk 2.0 or later.

Multi-PC real-time applications built before MotionDesk 2.0 do not have time stamps and therefore are not track safe. You have to open the models with a Release later than dSPACE Release 4.2 and in a MATLAB version later than MATLAB R13.0.1 to convert, rebuild, and download the models.

1-PC Solution All real-time applications assigned to dSPACE boards are track safe but without recording synchronization.

Applications built with RTI and MotionDesk Blockset 2.0 or later are also track-synchronized if you select the recording synchronization in the MD_Communication block. Refer to MD_Communication (MotionDesk Calculating and Streaming Motion Data 🕮).

Related topics

Basics

Basics of the Multi-Track Mode..... 109

Workflow for the Multi-Track Mode

Introduction

This topic gives an overview of the steps performed in the multi-track mode.

Workflow

If you want to compare a simulation with reference tracks, and you want synchronous replay, you must use the multi-track mode.

1. You have to create and add a track for each data source you want to replay. Refer to How to Create a Track on page 116.

   The multi-track mode is enabled automatically when you create a track.

   The number of tracks you can create is limited to 30. The more tracks you create, the slower the animation gets. If you want to make room for other tracks or improve the animation performance, you can delete tracks. You can delete them only individually. Refer to Delete (Track) on page 155.
2. When you create tracks, you define certain track properties. You can configure these properties later on, for example, to add a time offset for a recorded track or choose a new data source. Refer to How to Configure Track Properties on page 117.
3. If you want to compare simulations, you can run only one in an experiment. You have to replay the others as reference tracks. Refer to How to Compare a Simulation With Recorded Tracks on page 118.
4. Tracks become visible automatically when created. You can make them invisible individually if needed. Refer to Visible (Object) on page 170.

How to Create a Track

Objective

If you want to compare a simulation with reference tracks, and you want synchronous replay, you must use the multi-track mode. You have to create a track for each data source you want to replay and add it to the experiment.

Preconditions

An MDF file or a running simulation is available as the data source for the track.

Restrictions

You can create up to 30 tracks.

Tip

You can check how many frames you already used. The maximum is 132,000 frames. Refer to Customize Data Source on page 172.

Method

To create a track

1 On the **Scene Navigator**, select the **Tracks** folder. In the **Tracks'** context menu, select **Create Track**.

MotionDesk creates a new track.

2 Select the new track in the **Scene Navigator**.

The **Properties** pane displays the properties of the new track.

3 Specify the track's name in the **Name** field and select the parameters.

Specify a time offset if the data sources for the simulation and the track are identical in parts. The time offset helps you to distinguish them.

4 Under **Data source**, select **MDF file** and click the Browse button to select the file.

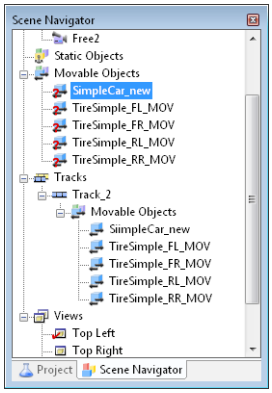
Or

Click **Fetch Simulation Data** to copy the data from the simulation buffer to the track buffer.

5 Click **OK** to confirm your settings.

Result

The track is created according to your settings. It is added to the experiment and its motion data is loaded.



Related topics

HowTos

How to Configure Track Properties..... 117

How to Configure Track Properties

Objective

When you create tracks, you define certain track properties. You can configure these properties later on, for example, to add a time offset or choose a new data source.

Method

To configure track properties

1 On the **Scene Navigator** under the **Tracks** node, click a track.

The **Properties** pane displays its properties.

2 Configure the track properties according to your needs. For details on the properties, refer to Track Properties on page 204.

3 Click **OK** to confirm.

Result

The properties are changed according to your alterations.

Related topics

HowTos

How to Create a Track 116

How to Compare a Simulation With Recorded Tracks

Objective

If you run a simulation and want to compare it with others, you have to use recorded tracks. Tracks use MDF files or the buffer of Simulink simulations or real-time applications as data sources.

When running the simulation and the tracks in multi-track mode, you can either observe them all in one window or observe the tracks in separate windows.

To distinguish movable objects of different tracks from each other, you can assign different colors to the objects. Refer to Object Properties (MotionDesk Scene Creation 🕮). You can also define a time offset to start the tracks with a delay. Refer to How to Configure Track Properties on page 117.

Preconditions

- All the required tracks are added.
- The simulation ran once to load the motion data.
- The **3-D View** is split into the desired number of views (maximum of four).
- The real-time application is running.

Method

To compare a simulation with recorded tracks

1 From a view's context menu, select the track that is to be displayed in this view.

The view shows the scene according to the selected track.

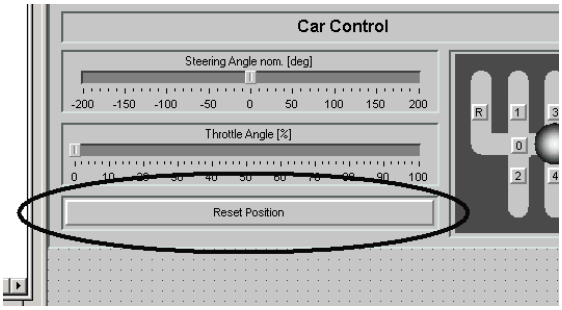
2 Open a view's context menu, select the observer you want to use to look at the scene in this view.

The view shows the scene from the selected observer's viewpoint.

3 Repeat the previous steps for all other views.

Each view shows the selected scene according to the selected track from the selected viewpoint.

4 Reset the experiment, for example, ControlDesk by clicking **Reset position** in the ControlDesk layout.



Result

MotionDesk clears the simulation buffer and restarts the tracks simultaneously with the simulation.

Related topics

HowTos

How to Adjust the 3-D View Windows 57
How to Create a Track 116
How to Replay an Animation 103

References

Observer Context 160

How to Hide or Show a Movable Object

Objective

Every movable object (original and clone) is made visible automatically when created. You can hide objects, for example, to improve the animation performance.

Method

To hide or show a movable object

1. On the **Scene Navigator**, select the movable object you want to hide or show.
2. In the object's context menu, click **Visible**.

Result

The movable object is hidden or shown.

Related topics

HowTos

How to Delete a Movable Object From a Track..... 120

How to Delete a Movable Object From a Track

Objective

If you do not need a movable object assigned to a track any more, you can delete it.

Method

To delete a movable object

1. On the **Scene Navigator**, select the movable object.
2. In the object's context menu, select **Delete**.

Tip

If you deleted a movable object of a track by mistake, you can restore it. Drag it from its original to the **Movable Objects** folder. You can create only a default clone. All previous modifications are lost.

Result

The movable object is deleted.

Related topics

HowTos

How to Hide or Show a Movable Object 120

Adapting the Animation

Introduction

You can use some tricks to improve the animation or performance.

Where to go from here

Information in this section

How to Select a Rendering Technique

Objective

The scene can be displayed in four different rendering techniques. Depending on the required authenticity, objects can be rendered in the wire frame or textures mode. For more information on the rendering techniques, refer to Rendering 3-D Graphics (MotionDesk Basics 🕮).

Possible methods

You can change the rendering techniques for the objects in the scene in two ways:

- Change the rendering technique for all objects in the scene in the same way. Refer to Method 1 on page 122.
- Change the rendering technique for individual objects in the scene. Refer to Method 2 on page 123.

Method 1

To change the rendering technique for the entire scene

1 On the Scene ribbon, click Scene – Render Mode and select the desired rendering technique.

Result

MotionDesk changes the rendering techniques for all objects of the scene in the same way.

Method 2

To change the rendering technique for individual objects

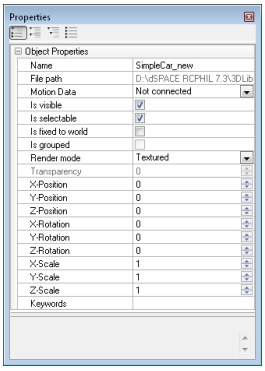
1 On the Scene ribbon, click Scene – Render Mode and select Mixed.

MotionDesk now displays each object in the rendering technique that is set in its properties. You can change these settings for each object individually.

2 On the Scene Navigator, select a static or movable object.

The Properties pane displays the properties of the selected object.

3 In the Properties pane, click in the value cell of the Render mode option.



4 From the list that opens, select the rendering technique.

5 Set the rendering techniques for each object in this way.

Result

MotionDesk changes the rendering techniques for each selected object individually.

Related topics

References

Render Mode..... 162

Getting Smooth Visualization

Introduction

Even if you use a high-end PC with a high-end graphics card, the frame rate can be slow, so the animation will be too. Here are some tips and tricks for improving the smoothness of the animation.

Use only one view	Displaying the scene from different viewpoints requires more processing power than displaying only one view because of the number of polygons displayed.
Disable shadow casting	Calculating shadows requires computation time and can reduce the frame rate. You can disable shadow casting for all or selected 3-D objects. For more information, refer to How to Specify Own Atmospheric Settings on page 131.
Reduce the display resolution	The rendering time depends on the number of pixels to be displayed. Thus reducing the display resolution, for example, to 1024 x 768 pixel, also reduces the rendering time. This is recommended for a very low latency time for man-in-the-loop simulation.
Use a simpler rendering technique	Rendering the objects with wire frames requires less processing power than rendering them with textures. You can choose the same rendering technique for the entire scene or individually select a rendering technique for each 3-D object. For more information, refer to How to Select a Rendering Technique on page 122.
Delete static objects	If your scene has a large number of static objects, you can delete some of them. For more information, refer to How to Delete Objects from the Scene (MotionDesk Scene Creation 🕮).
Reduce the number of polygons	If your scene has objects with a large number of polygons, you should use alternative objects with fewer polygons. For more information, refer to Editing a Scene in the 3-D View (MotionDesk Scene Creation 🕮).
Deactivate or reduce antialiasing and texture filtering	If antialiasing and texture filtering are activated for MotionDesk, you can deactivate them or reduce their quality settings. For more information, refer to How to Activate Anti-aliasing and Texture Filtering on page 125.
Related topics	HowTos How to Adjust the 3-D View Windows ..... 57 How to Get Information About a Scene 126

How to Activate Anti-aliasing and Texture Filtering

Objective

You can use the anti-aliasing and texture filtering functions of your graphics card to improve the quality of the graphical representation.

Anti-aliasing

Anti-aliasing is used to make edges appear smoother in a 3-D view.

Typically, the graphics card's driver allows you to change the anti-aliasing quality in steps such as Off, 2x, 4x, 8x, and 16x.

Texture filtering

Texture filtering is used to reduce the pixel noise that often appears when a finely structured texture is displayed in the distance. Pixel noise effects are particularly visible when the camera is moving in the 3-D View.

Typically, the graphics card's driver allows you to select a specific texture filtering function, such as anisotropic filtering, and to change its quality in steps such as Off, 2x, 4x, 8x, and 16x.

Optimum settings

There is a compromise between graphical quality and overall performance. For example, pushing up the graphical quality may slow down the frame rate and increase the latency.

The following settings are recommended:

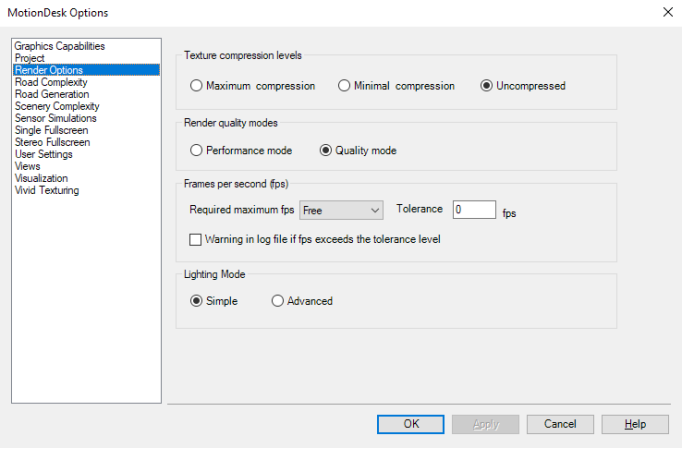
- Set anti-aliasing to 4x. A higher setting does not improve the quality significantly.
- Set texture filtering to 4x or higher. A lower setting may lead to annoying blur effects.

Method

To activate anti-aliasing and texture filtering

1. Open the Windows control panel of the graphics card.
2. Activate anti-aliasing and texture filtering on the hardware via the graphics card's control panel (which is usually a part of the advanced display properties). If the control panel has a setting called application controlled or similar, you must deactivate it.
3. Activate override application settings or similar and select the quality (4x or 8x, for example).

4 In MotionDesk on the File ribbon, click Options to open the MotionDesk Options dialog and change to the Render Options page.



5 In Render quality modes, select Quality mode to activate anti-aliasing and texture filtering.

Related topics

Basics

Getting Smooth Visualization 123

HowTos

How to Select a Rendering Technique 122

References

Render Options Page 208

How to Get Information About a Scene

Objective

MotionDesk provides information about the scene.

Method

To get information about a scene

1 On the Scene ribbon, click Scene - Scene Information.

MotionDesk displays the Scene Information dialog.

If an animation is running or you change the viewpoint, the number of visible polygons changes.

2 Click Update to get new values.

Result

MotionDesk displays information about the scene.

Related topics

References

Scene Information..... 163

Atmospheric Settings

Introduction

Setting the atmosphere makes your animation more realistic. You can specify the light and fog settings and reduce the visibility range to hide objects which are far away.

Where to go from here

Information in this section

Basics of Atmospheric Settings

Introduction

The scene looks more realistic if the atmosphere is included.

Limitations

The atmosphere is only visualized. There are the following limitations:

- There is no effect on the simulation model. For example, even if the scene looks hazy or foggy, the road is still dry.
- Collisions with 3-D objects in the scene are not detected.

Lights

Two different lights are used in the scene.

Headlight The observer light is fixed to the active observer. It is a point light which moves with the observer. In MotionDesk you can vary the intensity of the light but not the position or color.

Sunlight Sunlight lights the whole scene. The light source is far away, so the rays are parallel in the scene. In MotionDesk you can define the direction, color and intensity of the rays.

Shadows

For greater realism, you can enable the visualization of shadows in the scene. As this requires calculation power, you can disable it for all 3-D objects or some of them if the frame rate is too low.

Shadow casting is enabled by default. Shadow casting can only be disabled in the custom atmospherics mode.

Visibility range

Especially if your scene is very large, it is useful to reduce the visibility range. The lower the visibility range, the lower the number of objects which must be drawn and the higher the frame rate. The visibility range defines the distance from the observer to the clipping plane. For information on the clipping plane, refer to Viewpoints, Clipping Plane (MotionDesk Basics 🕮).

Fog

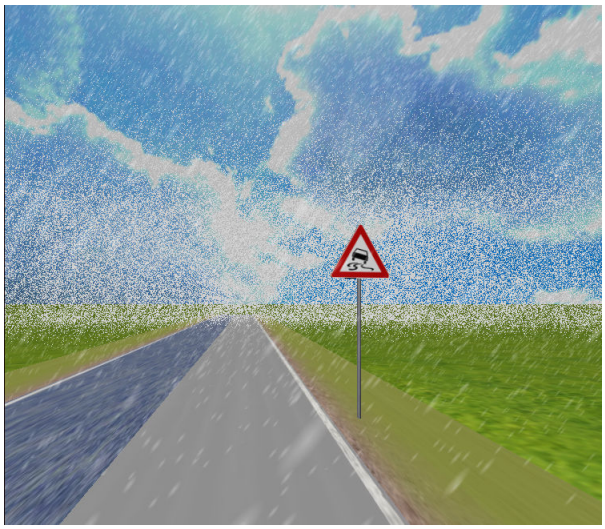
Fog reduces the visibility of objects. The greater the distance from the observer to the object, the lower the visibility of the objects. In MotionDesk you can define the color of the fog and how its density increases.

Tip

If you reduce the visibility range, you should also activate a fog to smoothly fade in the static objects when you move towards them. Otherwise, the clipping effect is clearly visible as static objects such as the horizon disappear and reappear abruptly.

Rain and snowfall

MotionDesk can visualize rain and snowfall. You can visualize only one of them or combine them.



Predefined atmosphere settings

MotionDesk provides predefined atmosphere settings, such as foggy, hazy, night, or sunset. These cannot be changed but you can use them as a basis for specifying your own settings.

Related topics

HowTos

How to Specify Own Atmospheric Settings 131

Examples

Example of Using Atmospherics Settings to Enhance the 3-D Performance of Large Scenes 133

How to Use Predefined Atmospherics Settings

Objective

You can use predefined atmosphere settings.

Atmosphere settings

MotionDesk provides several presets to specify the atmosphere. These are the simplest way to set the atmospherics.

For details on atmospherics settings, refer to Basics of Atmospheric Settings on page 128.

Method

To use a predefined atmospherics settings

1 On the Scene ribbon, click Scene – Atmospherics and select a predefined setting.

Result

The atmosphere of the scene is set to the selected settings.

Related topics

References

Atmospherics Mode..... 142

How to Specify Own Atmospheric Settings

Objective

You can define your own atmospheric settings on the basis of preset atmosphere setting.

Atmospheric settings

MotionDesk provides several presets to specify the atmospherics. The simplest way is to use these settings for your visualization. You can also use them as templates and adapt their property values to your requirements.

For more details, refer to Basics of Atmospheric Settings on page 128.

Method

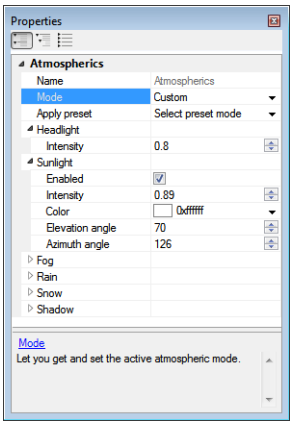
To specify own atmospheric settings

1 Open the Scene Navigator and Properties pane.

2 In the Scene Navigator, select the Atmospherics node.

The Properties pane displays the properties for specifying the atmospheric settings.

3 On the Properties pane in the Mode property, select Custom.



4 In the Apply preset property, select a preset mode.

The atmosphere of the scene is set to the selected atmospherics. You can use the settings of the properties as start values for specifying your own atmosphere settings.

5 On the Properties pane, edit the properties as required. For a description of the properties, refer to Atmospherics Properties on page 189.

Result

The atmosphere of the scene is set.

Related topics

Examples

Example of Using Atmospherics Settings to Enhance the 3-D Performance of Large Scenes 133

How to Enable or Disable Shadow Casting

Objective

Shadow casting can be enabled or disabled for single or all objects.

Calculating the shadows requires computation time and may reduce the frame rate. If the frame rate becomes too slow, you can disable shadow calculation for some or all 3-D objects.

By default, shadow casting is enabled. Shadow casting can only be disabled in the custom atmospherics mode.

Preconditions

The custom mode is set for atmospherics.

Possible methods

You can enable or disable shadow casting:

- For a single object. Refer to Method 1 on page 133.
- For a group of objects. Refer to Method 2 on page 133.

Method 1

To enable or disable shadow casting for a single object

1 In the Scene Navigator, select the object.

2 Open the object's context menu and choose Enable Shadows or Disable Shadows

You can also select or clear the Cast shadow property on the Properties pane.

Method 2

To enable or disable shadow casting for a group of objects

1 In the Scene Navigator, select the Static Objects, group, or Movable Objects node.

2 Open the context menu and select Enable Shadows or Disable Shadows.

Result

You can see shadows of the 3-D objects whose shadow casting is enabled.

Tip

The shadows depend on the properties of the sunlight in the scene. For details on modifying the sunlight's properties, refer to How to Specify Own Atmospheric Settings on page 131.

Related topics

References

Example of Using Atmospherics Settings to Enhance the 3-D Performance of Large Scenes

Introduction

The example shows how you can set the atmospherics for a large scene. If your scene is very large, many objects may be in the visibility range even if they are far away. The larger the number of objects to be drawn, the lower the frame rate. Using atmospherics settings, you can reduce the visibility range to hide objects and raise the frame rate.

Reducing the visibility range

To reduce the visibility range, specify the following values for atmospherics:

Category	Property	Value
Fog	Enabled	Checked
	Color	Light blue
	Range	200 m
	Density	Exponential Squared

With these settings, every object which is more than 200 m away from the observer is hidden. Because of the light-blue color of the fog, the background looks like the sky, see the following illustration.

Related topics

Basics

Basics of Atmospheric Settings..... 128

HowTos

How to Specify Own Atmospheric Settings..... 131

References

Atmospherics Properties..... 189

Reference Information

Where to go from here

Information in this section

Scene Manager Commands

Introduction

The Scene Manager provides various commands, which are accessible via the ribbon and the context menus.

Where to go from here

Information in this section

Active Observer

Access

You can access this command via:

Ribbon	Observation – Observer
Context menu of	3-D View
Shortcut key	None
Icon	None

Purpose

To select the observer to be used in the active 3-D View window.

Result

The selected observer is used.

Related topics

Basics

Basics of Observers and the ModelDesk Observer 79
Default Observers 81

HowTos

How to Select an Observer 84

References

Create Default Observers 147

Activate Shadows

Access

You can access this command via:

Ribbon	None
Context menu of	▪ Static Objects in Scene Navigator ▪ Movable Objects in Scene Navigator
Shortcut key	None
Icon	None

Purpose

To activate shadow casting for the selected 3-D objects in the animation.

Result

The shadows of the selected objects are calculated and displayed.

Description

You can activate the calculation and casting of shadows for more realistic visualization. You can do this for each 3-D object individually or for groups of 3-D objects. To activate it for one 3-D object, open its context menu in the **Scene Navigator** and select **Activate shadows**. To activate it for a group of 3-D objects, open the context menu of the group, for example, **Static objects**, and select **Activate shadows**. You must also activate shadow calculation globally with an Atmospherics property. Refer to Atmospherics Properties on page 189.

To calculate the shadows, the properties of the sunlight specification are used, especially the elevation and azimuth angles. Refer to Atmospherics Properties on page 189.

Related topics

HowTos

How to Specify Own Atmospheric Settings..... 131

References

Deactivate Shadows..... 156

Add to Panel

Access

You can access this command via:

Ribbon	None
Context menu of	Instrument in the **Scene Navigator**
Shortcut key	None
Icon	None

Purpose

To add an instrument to an Instrument Panel.

Result

The selected instrument is moved to an Instrument Panel.

Description

When you use the **Add to Panel** command, the **Select Panel** dialog opens for you to select an Instrument Panel. When it is selected, the instrument becomes a child instrument of this panel. You can modify the instrument's properties to specify the position relative to the panel. Refer to Instrument Properties on page 192.

It is not possible to cascade Instrument Panels. You cannot use an Instrument Panel as child of another Instrument Panel.

Tip

Alternative: To add an instrument, you can drag a stand-alone instrument to an Instrument Panel in the **Scene Navigator**.

Related topics

References

Remove from Panel..... 161

Align Free Observer

Access

You can access this command via:

Ribbon	▪ Scene – Observer ▪ Observation – Observer
Context menu of	None
Shortcut key	None
Icon	

Purpose

To align the view of a free observer parallel to the base plate.

Result

The roll and pitch of the active view are set to zero. In other words, the view is parallel to the base plate through all angles of azimuth.

Description

The roll and pitch of observers that are defined as *Free* can be altered freely using the mouse in conjunction with the **Tab** key. If when altering the observer view you lose your orientation in the virtual world, you can quickly realign the view with this command.

Related topics

Basics

Basics of Observers and the ModelDesk Observer 79
Working with Observers in Zone Navigation Mode 96

Always on Top

Access

You can access this command via:

Ribbon	View – Full Screen
Context menu of	None
Shortcut key	None
Icon	

Purpose

To keep the MotionDesk window always on top of the screen.

Description

If this option is enabled, the window of MotionDesk is kept on the top of the display.

Related topics

HowTos

How to Configure a Stereo Display..... 76

References

Stereo Fullscreen Options Page..... 212

Atmospherics Mode

Access

You can access this command via:

Ribbon	Scene – Scene
Context menu of	None
Shortcut key	None
Icon	None

Purpose

To set the atmospherics for a specific environment.

Description

You can select one of the defined atmospherics settings (**Sunny**, **Hazy**, **Foggy**, **Sunset**, **Night**, **Rainy**, or **Snowy**) or specify a custom setting. The defined atmospherics settings cannot be modified.

To specify your own atmospherics settings, select **Custom** and modify the properties on the **Properties** pane. For a description of the properties, refer to Atmospherics Properties on page 189.

Result

The scene environment is set to the selected atmospherics.

Related topics

HowTos

How to Specify Own Atmospheric Settings..... 131

Bring to Front

Access

You can access this command via:

Ribbon	Observation - Layer Instrument
Context menu of	Instrument in the Scene Navigator
Shortcut key	None
Icon	

Purpose

To bring the selected instrument to the front.

Result

The selected instrument will cover the instruments in the background if their borders overlapped before this command was carried out.

Related topics

HowTos

How to Use Instruments in the Scene 33

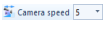
References

Move Layer Down 159
Move Layer Up 159
Send to Back 164

Camera Speed (Observer)

Access

You can access this command via:

Ribbon	Observation – Camera speed
Context menu of	None
Shortcut key	None
Icon	

Purpose

Select Observer - Camera speed to adjust the speed at which you move through the scene.

Description

You can select the **Observer navigation** camera speed to speed up or slow down the speed at which you move through the scene. The range is from 1 to 10.

Camera speed cannot be selected for zone navigation.

Related topics

Basics

Overview of the Observer Navigation Mode....85

HowTos

How to Select the Observer Navigation Mode and Speed....87

Create (Instrument)

Access

You can access this command via:

Ribbon	None
Context menu of	**Instruments** folder in **Scene Navigator**
Shortcut key	None
Icon	

Purpose

To create a new instrument and add it to the experiment.

Description

An instrument is created with default properties. You can modify the properties on the **Properties** pane, for example, changing the instrument type.

You can also directly add an instrument of a specific type via a ribbon command:

- **Create Numeric Instrument**
- **Create Gauge Instrument**
- **Create Bar Instrument**
- **Create LED Instrument**
- **Create Picture Instrument**
- **Create Text Instrument**

Result

A new instrument is created and added to the experiment.

Related topics

Basics

Basics on Using Instruments in the Scene....31

HowTos

How to Use Instruments in the Scene....33

References

Delete (Instrument/Instrument Panel)....154
Instrument Properties....192
Visible (Instrument/Instrument Panel)....169

Create (Observer)

Access

You can access this command via:

Ribbon	None
Context menu of	Observers in Scene Navigator
Shortcut key	None
Icon	

Purpose

To create a new observer.

Description

The properties of the observer are specified in the Observer Properties dialog. Refer to Observer Properties on page 200.

Result

A new observer is created and added to the experiment.

Related topics

Basics

Working with Observers 79

HowTos

How to Create and Configure a Custom Observer 82

References

Create Default Observers 147
Delete (Observer) 155
Observer Properties 200

Create Bar Instrument

Access

You can access this command via:

Ribbon	Observation – Create Instrument
Context menu of	None
Shortcut key	None
Icon	

Purpose

To create a new instrument of the Bar type.

Description

An instrument of the Bar type is created with default properties. You can modify the properties on the Properties pane.

Result

A new instrument is created and added to the experiment.

Related topics

Basics

Basics on Using Instruments in the Scene....31

HowTos

How to Use Instruments in the Scene....33

References

Instrument Properties....192

Create Default Observers

Access

You can access this command via:

Ribbon	▪ Scene – Observer ▪ Observation – Observer
Context menu of	None
Shortcut key	None
Icon	

Purpose

To create a new series of default observers.

Result

The new observers appear in the Scene Navigator under Observers.

Description

The Configure Experiment dialog opens. In this dialog you select a movable object in the scene to attach the new default observers to.

If you click OK in the dialog, MotionDesk creates a new series of observers which are attached to the selected movable object. You can use this function more than once, for example, to create a set of observers that follow a second car in the scene.

If your experiment does not have a movable object, you can cancel the dialog and add an object in 3-D View.

Related topics

HowTos

How to Create Default Observers 81

References

Active Observer 138
Create (Observer) 145
Delete (Observer) 155

Create Gauge Instrument

Access

You can access this command via:

Ribbon	Observation – Create Instrument
Context menu of	None
Shortcut key	None
Icon	

Purpose

To create a new instrument of the Gauge type.

Description

An instrument of the Gauge type is created with default properties. You can modify the properties on the Properties pane.

Result

A new instrument is created and added to the experiment.

Related topics

Basics

Basics on Using Instruments in the Scene 31

HowTos

How to Use Instruments in the Scene 33

References

Instrument Properties 192

Create LED Instrument

Access

You can access this command via:

Ribbon	Observation – Create Instrument
Context menu of	None
Shortcut key	None
Icon	

Purpose

To create a new instrument of the Multistate LED type.

Description

An instrument of the Multistate LED type is created with default properties. You can modify the properties on the **Properties** pane.

Result

A new instrument is created and added to the experiment.

Related topics

Basics

Basics on Using Instruments in the Scene..... 31

HowTos

How to Use Instruments in the Scene..... 33

References

Instrument Properties..... 192

Create Numeric Instrument

Access

You can access this command via:

Ribbon	Observation – Create Instrument
Context menu of	None
Shortcut key	None
Icon	

Purpose	To create a new instrument of the Numerics type.
Description	An instrument of the Numerics type is created with default properties. You can modify the properties on the **Properties** pane.
Result	A new instrument is created and added to the experiment.
Related topics	Basics Basics on Using Instruments in the Scene 31 HowTos How to Use Instruments in the Scene 33 References Instrument Properties 192

Create Panel

Access

You can access this command via:

Ribbon	**Observation – Create Instrument**
Context menu of	**Instruments in Scene Navigator**
Shortcut key	None
Icon	

Purpose	To create an Instrument Panel.
Description	An Instrument Panel is created with default properties. You can modify the properties on the **Properties** pane. Refer to Instrument Properties on page 192.
Result	A new Instrument Panel is created and added to the experiment.

Related topics

Basics

Basics on Using Instrument Panels in the Scene 36

HowTos

How to Use Instrument Panels in the Scene 37

Create Picture Instrument

Access

You can access this command via:

Ribbon	Observation – Create Instrument
Context menu of	None
Shortcut key	None
Icon	

Purpose

To create a new instrument of the Multistate Picture type.

Description

An instrument of the Multistate Picture type is created with default properties. You can modify the properties on the **Properties** pane.

Result

A new instrument is created and added to the experiment.

Related topics

Basics

Basics on Using Instruments in the Scene 31

HowTos

How to Use Instruments in the Scene 33

References

Instrument Properties 192

Create Text Instrument

Access

You can access this command via:

Ribbon	Observation – Create Instrument
Context menu of	None
Shortcut key	None
Icon	

Purpose

To create a new instrument of the Multistate Text type.

Description

An instrument of the Multistate Text type is created with default properties. You can modify the properties on the Properties pane.

Result

A new instrument is created and added to the experiment.

Related topics

Basics

Basics on Using Instruments in the Scene 31

HowTos

How to Use Instruments in the Scene 33

References

Instrument Properties 192

Create Track

Access

You can access this command via:

Ribbon	None
Context menu of	Tracks folder in Scene Navigator
Shortcut key	None
Icon	

Purpose	To create a new track and add it to the experiment.
Description	A track is created with default properties. You can modify the properties on the Properties pane. For details, refer to Track Properties on page 204.
Result	A new track is created and added to the experiment.
Related topics	References Delete (Track) 155

Data Stream Selector

Access	You can hide or show the Data Stream Selector via View - Controlbar - Switch Controlbars.
Purpose	To connect or disconnect a data stream to a movable object or instrument in the Scene Navigator via drag & drop.
Result	The data stream that is dropped onto a movable object or instrument is assigned to this object. If the disconnect item is dropped onto a movable object or instrument, the assignment of this object is deleted.
Description	The Data Stream Selector lists all data streams that are embedded in the simulation model. The Scene Navigator lists all movable objects that are used in the scene. Via drag & drop you can connect data streams to movable objects.
Dialog settings	**Data stream list** Lists the data streams available in the model as well as the disconnect item.

Related topics

Basics

Using Instruments in the Scene....31

HowTos

How to Assign Motion Data to Movable Objects....26

Delete (Instrument/Instrument Panel)

Access

You can access this command via:

Ribbon	None
Context menu of	Instrument or Instrument Panel in Scene Navigator
Shortcut key	**Del**
Icon	

Purpose

To delete the instrument or Instrument Panel selected in the Scene Navigator.

Description

If you do not need an instrument or Instrument Panel anymore, you can delete it to make room for other instruments or to improve the animation speed.

To delete several instruments or Instrument Panels, select them in the Scene Navigator using multiselection and press **Del**.

Result

The selected instrument or Instrument Panel is removed from the Instruments folder in the Scene Navigator.

Related topics

HowTos

How to Use Instrument Panels in the Scene....37

Delete (Observer)

Access

You can access this command via:

Ribbon	None
Context menu of	Observer in **Scene Navigator**
Shortcut key	None
Icon	

Purpose

To delete the observer selected in the **Scene Navigator**.

Description

If you delete an observer that is currently assigned to one of the views displayed in the **3-D View**, the view is automatically assigned to the **Free1** observer.

You cannot delete the **Free1** and **Free2** observers.

Result

The selected observer is removed from the Observers folder in the **Scene Navigator**.

Related topics

References

Create (Observer) 145
Create Default Observers 147

Delete (Track)

Access

You can access this command via:

Ribbon	None
Context menu of	Track in **Scene Navigator**
Shortcut key	None
Icon	

Purpose

To delete the track selected in the **Scene Navigator**.

Result

The selected track is removed from the list of tracks in the **Scene Navigator**.

Description

If you do not need a track anymore, you can delete it to make room for other tracks or to improve the animation speed. You can delete tracks only individually.

Related topics

References

Create Track..... 152

Deactivate Shadows

Access

You can access this command via:

Ribbon	None
Context menu of	▪ **Static Objects** in **Scene Navigator** ▪ **Movable Objects** in **Scene Navigator**
Shortcut key	None
Icon	None

Purpose

To deactivate shadow casting for 3-D objects in the scene.

Result

The shadows of the selected objects are neither calculated nor cast.

Description

If the frame rate of the animation is too slow, you can deactivate the calculation and casting of shadows of 3-D objects in the scene. You can do this for each 3-D object individually or for groups of 3-D objects. To disable it for one 3-D object, open its context menu in the **Scene Navigator** and select **Deactivate shadows**. To deactivate it for a group of 3-D objects, open the context menu of the group, for example, **Static objects**, and select **Deactivate shadows**.

Related topics

HowTos

How to Specify Own Atmospheric Settings..... 131

References

Activate Shadows..... 139

Four Views

Access

You can access this command via:

Ribbon	▪ Observation – Views ▪ Home – Views
Context menu of	None
Shortcut key	**CTRL+4**
Icon	

Purpose

To display four views in the 3-D View.

Result

The 3-D View displays four views.

Description

The 3-D View displays the views as seen from the observers in the experiment. A red check mark next to an observer in the Scene Navigator indicates which observer is currently selected. The active view, which is indicated by a red frame around the view, displays the view of the currently selected observer.

The views are arranged so that each view initially has the same size. You can alter their sizes by dragging the frames.

By double-clicking one of the views, you can switch to the one view mode.

Note

If you display more than one view, the frame rate reduces.

Related topics

HowTos

How to Adjust the 3-D View Windows 57

References

One View 161
Tile Horizontally 166
Tile Vertically 167

Make Primary

Access

You can access this command via:

Ribbon	None
Context menu of	Child instrument in **Scene Navigator**
Shortcut key	None
Icon	None

Purpose

To define a child instrument as the primary child instrument.

Result

The selected instrument is the primary child instrument.

Description

You can define one child instrument of an Instrument Panel as the primary child instrument. The values of the parameter that is connected to the primary child instrument can be visualized in the title bar of the minimized Instrument Panel. Refer to How to Configure the Title Bar of a Minimized Instrument Panel on page 40.

Related topics

HowTos

How to Use Instrument Panels in the Scene 37

Move Layer Down

Access

You can access this command via:

Ribbon	Observation – Layer Instrument
Context menu of	Instrument in the Scene Navigator
Shortcut key	None
Icon	

Purpose

To move the selected instrument in the layout one layer down.

Result

The selected instrument is covered by the instruments on higher layers if their borders overlapped before this command was carried out.

Related topics

HowTos

How to Use Instruments in the Scene 33

References

Bring to Front 143
Move Layer Up 159
Send to Back 164

Move Layer Up

Access

You can access this command via:

Ribbon	Observation – Layer Instrument
Context menu of	Instrument in the Scene Navigator
Shortcut key	None
Icon	

Purpose

To move the selected instrument in the layout one layer up.

Result

The selected instrument will cover the instrument on lower layers if their borders overlapped before this command was carried out.

Related topics

HowTos

How to Use Instruments in the Scene..... 33

References

Bring to Front..... 143
Move Layer Down..... 159
Send to Back..... 164

Observer Context

Access

You can access this command via:

Ribbon	Observation – Observer
Context menu of	None
Shortcut key	None
Icon	None

Purpose

To select the context of the observer in the active view.

Result

The observer follows the object moved in the simulation or a recorded track.

Description

If you use the multi-track mode to compare the simulation with recorded tracks, you can use **Observer Context** to select whether the observer follows the object moved with the simulation data or a recorded track.

Related topics

Basics

Workflow for the Multi-Track Mode..... 115

HowTos

How to Compare a Simulation With Recorded Tracks..... 118

One View

Access

You can access this command via:

Ribbon	▪ Observation – Views ▪ Home - Views
Context menu of	None
Shortcut key	**CTRL+1**
Icon	

Purpose

To display the view from the observer currently selected in the 3-D View in one window.

Description

The 3-D View displays the views as seen from the observers in the experiment. A red check mark next to an observer in the Scene Navigator indicates which observer is currently selected.

Related topics

HowTos

How to Adjust the 3-D View Windows 57

References

Four Views 157
Tile Horizontally 166
Tile Vertically 167

Remove from Panel

Access

You can access this command via:

Ribbon	None
Context menu of	Instrument in an Instrument Panel in the Scene Navigator
Shortcut key	None
Icon	None

Purpose

To remove an instrument from an Instrument Panel.

Result

The selected instrument is removed from the Instrument Panel and appears as a stand-alone instrument.

Description

When you use the **Remove from Panel** command, the selected instrument is only removed from the panel but not deleted. It remains in the scene with the same settings.

Tip

Alternative: To remove a child instrument, you can drag the child instrument from the Instrument Panel to the Instruments node in the **Scene Navigator**.

Related topics

References

Add to Panel..... 140
Delete (Instrument/Instrument Panel)..... 154

Render Mode

Access

You can access this command via:

Ribbon	Scene – Scene
Context menu of	None
Shortcut key	None
Icon	

Purpose

To select the render mode for each object in the scene.

Description

You can select the render mode for each object individually by setting the **Render mode** property. However, the render mode selected in the **Render Mode** selection box is set for the entire scene. The individually assigned render mode settings remain defined for each object.

Render Mode	Description
Mixed	Lets you change the rendering technique for individual objects.
Textured	Rendering with Gouraud shading and texture mapping.
Wireframe	Rendering with wire frames

Related topics

Basics

Rendering 3-D Objects with Texture Mapping (MotionDesk Basics 🕮)
Rendering 3-D Objects with Wire Frames (MotionDesk Basics 🕮)

HowTos

How to Select a Rendering Technique..... 122

Scene Information

Access

You can access this command via:

Ribbon	Scene - Scene
Context menu of	None
Shortcut key	None
Icon	

Purpose

To display information about the loaded scene.

Result

Information about the number of polygons used and the texture memory consumption is displayed.

Scene Information dialog

The Scene Information dialog displays:

- Number of movable and static objects of the scene
- Number of polygons used by movable and static objects
- Size of used texture memory
- Number of objects that are visible in the views

Update Updates the specified information. This is useful if you change the observer position in a view or switch to a different view mode, or if an animation is running.

Related topics

HowTos

How to Get Information About a Scene..... 126

Scene Navigator

Access

You can access the Scene Navigator via:

Ribbon	View – Controlbar – Switch Controlbars
Context menu of	None
Shortcut key	`Ctrl+Alt+3`
Icon	None

Purpose

To access the elements of a scene.

Result

Depending on the type of the element, its properties are displayed in the Properties, and/or its context menu provides commands for working with it.

Description

The Scene Navigator lists all the elements that belong to the scene. The elements are structured in a tree view.

Send to Back

Access

You can access this command via:

Ribbon	Observation – Layer Instrument
Context menu of	Instrument in the Scene Navigator
Shortcut key	None
Icon	

Purpose

To send the selected instrument to the back.

Result

The selected instrument is covered by the instruments in the foreground if their borders overlapped before this command was carried out.

Related topics

HowTos

How to Use Instruments in the Scene....33

References

Bring to Front....143
Move Layer Down....159
Move Layer Up....159

Single Full Screen

Access

You can access this command via:

Ribbon	▪ View - Full Screen ▪ Home - Views
Context menu of	None
Shortcut key	**F11**
Icon	

Purpose

To activate the full screen mode.

Description

In the full screen mode, the scene is displayed across several displays as it is configured on the **Single Fullscreen** page.

To return to the normal view, press **Esc** or **F11**.

Related topics

HowTos

How to Configure Full Screen Mode with Multiple Displays....73

References

Single Fullscreen Options Page....210

Stereo Full Screen

Access

You can access this command via:

Ribbon	View – Full Screen
Context menu of	None
Shortcut key	**F12**
Icon	

Purpose

To toggle between stereo and normal view.

Description

To display the scene in stereo, two displays are required. One display shows the scene for the left eye, a second display shows the scene for the right eye. Both displays show the scene in fullscreen. The stereo mode is configured on the Stereo Fullscreen page.

To return to the normal view, press **Esc** or **F12**.

Related topics

HowTos

How to Configure a Stereo Display 76

References

Stereo Fullscreen Options Page 212

Tile Horizontally

Access

You can access this command via:

Ribbon	View – Window
Context menu of	None
Shortcut key	**CTRL+2**
Icon	

Purpose

To display two views arranged horizontally in the 3-D View.

Result

The 3-D View displays two views and arranges them horizontally.

Description

The 3-D View displays the views as seen from the observers in the experiment. A red check mark next to an observer in the Scene Navigator indicates which observer is currently selected. The active view, which is indicated by a red frame, displays the view of the currently selected observer.

The views are arranged so that they are all initially the same size. You can alter their sizes by dragging the frames.

By double-clicking one of the views, you can switch to the one view mode.

Note

If you display more than one view, the frame rate reduces.

Related topics

HowTos

How to Adjust the 3-D View Windows 57

References

Four Views 157
One View 161
Tile Vertically 167

Tile Vertically

Access

You can access this command via:

Ribbon	View – Window
Context menu of	None
Shortcut key	CTRL+3
Icon	

Purpose

To display two views arranged vertically in the 3-D View.

Result

The 3-D View displays two views and arranges them vertically.

Description

The 3-D View displays the views as seen from the observers in the experiment. A red check mark next to an observer in the Scene Navigator indicates which observer is currently selected. The active view, which is indicated by a red frame, displays the view of the currently selected observer.

The views are arranged so that they are all initially the same size. You can alter their sizes by dragging the frames.

By double-clicking one of the views, you can switch to the one view mode.

Note

If you display more than one view, the frame rate reduces.

Related topics

HowTos

How to Adjust the 3-D View Windows 57

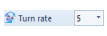
References

Four Views 157
One View 161
Tile Horizontally 166

Turn rate (Observer)

Access

You can access this command via:

Ribbon	Observation – Turn rate
Context menu of	None
Shortcut key	None
Icon	

Purpose

Select Observer - Turn rate to adjust the rotation speed of the observer on a fixed position to look around the scene.

Description

You can select the Observer navigation turn rate to speed up or slow down the speed at which you rotate on a fixed position to look around the scene. The range is from 1 to 10.

Turn rate cannot be selected for zone navigation.

Related topics

Basics

Overview of the Observer Navigation Mode....85

HowTos

How to Select the Observer Navigation Mode and Speed....87

Visible (Instrument/Instrument Panel)

Access

You can access this command via:

Ribbon	None
Context menu of	Instruments folder in **Scene Navigator**
Shortcut key	None
Icon	None
Others	**Instrument/Instrument Panel Properties**

Purpose

To make an instrument or Instrument Panel visible or invisible, respectively.

Description

Instruments or Instrument Panel become visible automatically when created. You can make them invisible individually if needed.

Result

The selected instrument or Instrument Panel is visible or invisible in the scene.

Related topics

HowTos

How to Use Instrument Panels in the Scene....37

Visible (Object)

Access

You can access this command via:

Ribbon	None
Context menu of	Object in **Scene Navigator**
Shortcut key	None
Icon	None
Others	**Object Properties**

Purpose

To make an object visible or invisible.

Result

The selected object is visible or invisible in the scene.

Description

Objects become visible automatically when created. You can make them invisible individually if needed.

Zone and Observer navigation

Access

You can access this command via:

Ribbon	**Observation – Zone navigation**
Context menu of	None
Shortcut key	None
Icon	

Purpose

To switch from **Observer navigation** to **Zone navigation**.

Description

When the icon is shown in an orange box, you have selected **Zone navigation**. You can click to reselect **Observer navigation**.

When the icon is shown without an orange box, you have selected **Observer navigation**.

Related topics

Basics

Overview of the Observer Navigation Mode....85

HowTos

How to Select the Observer Navigation Mode and Speed....87

Simulation Control and Motion Diagnostics

Introduction

The Simulation Control provides various commands and dialogs, which are accessible via the ribbon and the context menus of simulation-related components.

Where to go from here

Information in this section

Customize Data Source

Access

You can access this command via:

Ribbon	Home – Platform
Context menu of	None
Shortcut key	**CTRL+U**
Icon	▪ … in the Motion Player ▪ 

Purpose	To view or edit the data source properties.
Result	The Customize Data Source dialog opens to let you view or edit the data source properties.
Description	Several property pages are available. One is valid for all data sources, and the others are valid for specific data sources. All pages are available in the dialog if you access the command via the menu bar or the shortcut keys. If you access the command via the Browse button in the Motion Player, only the specific property page of the selected data source opens. You cannot access the file page via the Browse button. The command is disabled if None is selected as the data source.
General page	This page is valid for all data sources. It lets you specify the maximum number of simulation and track frames. A total of 132,000 frames is possible. **Max simulation frames** Lets you specify the maximum number of simulation frames. **Max track frames** Specifies the maximum number of frames of all tracks. This number adapts to changes to the maximum simulation frames so the two numbers do not exceed a total of 132,000 frames altogether. **Total track frames** Displays the number of frames that you already used in tracks. **Start MotionDesk animation automatically** Lets you start the animation in MotionDesk when the real-time simulation is started. This is only supported for platforms that are connected to ControlDesk, ModelDesk, or AutomationDesk. **Display a warning if frames get lost during data acquisition** Lets you enable a warning message that is displayed if the number of lost frames increases the value specified in Tolerance value. **Tolerance value** Lets you specify the number of frames which have been lost before a warning is displayed.
ADAS page	This page is valid only for a TCP ADAS simulator as the data source. It lets you choose an ADAS source on the network to provide the motion data for the animation. **Simulator IP address** Lets you select an ADAS simulator as the data source. Specify the TCP/IP address of the simulator found on the network. **Motion data port** Specify the motion data port number for the data source on the simulator on the network.

DS1007 page

This page is valid only for DS1007 PPC Processor Board as the data source. It lets you choose a DS1007 to provide the motion data for the animation.

Board Displays the information on the selected DS1007 (name and IP address).

Select Board Lets you select a DS1007 as the data source. If you click the button, a dialog opens that lists all the DS1007 boards found in the network. The list contains the board name, IP address, and serial number.

To select a board, click its entry and then OK.

To update the list, click Update.

Unassign Board Lets you unassign the DS1007 so that it is no longer used as a data source. If you do not unassign the board, no other user can access the simulation, unless the broadcast mode is used for the model.

Application Info Lets you select the real-time application to provide the motion data. All real-time applications are listed that run on the DS1007 and include the MotionDesk services. When you select a real-time application, the values of Name, Path, Owner, and State are updated.

Property	Description
Name	Displays the name of the selected real-time application.
Path	Displays the path of the selected real-time application.
Owner	Displays the owner of the selected real-time application.
State	Displays the state of the selected real-time application.

File page

This page is valid only for MDF files as the data source. It lets you choose the MDF file (motion data) that is to be used for the animation.

Stream Displays the path and name of the currently selected MDF file.

You can use the Browse button [...] to select an MDF file. The chosen file appears as one of the available data sources in Motion Player.

Start frame Displays the number of the buffer's start frame.

End frame Displays the number of the buffer's end frame.

Buffer size Displays the number of frames that are buffered during the animation.

Hardware page

This page is valid only for the DS1006 processor board connected via a bus connection. The real-time board must be registered.

Stream Lets you select the real-time board that is used as data source.

Application Displays the path and name of the application running on the selected board.

Frame buffer size Displays the number of frames that are buffered during the animation.

Stream buffer size Lets you specify the number of frames that are buffered by streaming. You should specify this number if the number of simulation frames is very high and buffering the whole simulation would take too long.

MotionDesk data acquisition refresh interval Lets you specify a time interval during which MotionDesk reads the data from the hardware.

Hardware-Net page

This page is valid only for real-time boards connected via a network connection. It lets you select real-time boards such as MicroAutoBox II or a modular hardware connected via a slot CPU. The real-time boards must be registered.

Note

If a real-time board is connected via a network connection, the frame rate can be slow.

Stream Lets you select the real-time board that is used as data source.

Application Displays the path and name of the application running on the selected real-time board.

Frame buffer size Displays the number of frames that are buffered during the animation.

Stream buffer size Lets you specify the number of frames that are buffered by streaming. You should specify this number if the number of simulation frames is very high and buffering the whole simulation would take too long.

MicroAutoBox III page

This page is valid only for MicroAutoBox III as the data source. It lets you choose a MicroAutoBox III to provide the motion data for the animation.

Board Displays the information on the selected MicroAutoBox III (name and IP address).

Select Board Lets you select a MicroAutoBox III as the data source. If you click the button, a dialog opens that lists all MicroAutoBox IIIs, found in the network. The list contains the board name, IP address, and serial number.

To select a board, click its entry and then OK.

To update the list, click Update.

Unassign Board Lets you unassign MicroAutoBox III so that it is no longer used as a data source. If you do not unassign the board, no other user can access the simulation, unless broadcast mode is used for the model.

Application Info Lets you select the real-time application to provide the motion data. All real-time applications are listed that run on MicroAutoBox III and include the MotionDesk services. When you select a real-time application, the values of Name, Path, Owner, and State are updated.

Property	Description
Name	Displays the name of the selected real-time application.
Path	Displays the path of the selected real-time application.
Owner	Displays the owner of the selected real-time application.
State	Displays the state of the selected real-time application.

MLBX page

This page is valid only for MicroLabBox as the data source. It lets you choose a MicroLabBox to provide the motion data for the animation.

Board Displays the information on the selected MicroLabBox (name and IP address).

Select Board Lets you select a MicroLabBox as the data source. If you click the button, a dialog opens that lists all MicroLabBoxes, found in the network. The list contains the board name, IP address, and serial number.

To select a board, click its entry and then OK.

To update the list, click Update.

Unassign Board Lets you unassign MicroLabBox so that it is no longer used as a data source. If you do not unassign the board, no other user can access the simulation, unless broadcast mode is used for the model.

Application Info Lets you select the real-time application to provide the motion data. All real-time applications are listed that run on MicroLabBox and include the MotionDesk services. When you select a real-time application, the values of Name, Path, Owner, and State are updated.

Property	Description
Name	Displays the name of the selected real-time application.
Path	Displays the path of the selected real-time application.
Owner	Displays the owner of the selected real-time application.
State	Displays the state of the selected real-time application.

Network page

This page is valid only for network adapters as the data source. It lets you specify the settings of the network adapter that is to provide the motion data for the animation.

Edit Network Settings Indicates whether the network settings are enabled for editing.

Port #1 Lets you specify the UDP port number for the port that is used to send motion data from the simulator to the MotionDesk PCs. Each port must have a unique number between 1025 and 65535.

Port #2 Lets you specify the UDP port number for the port that is used to send configuration requests from the MotionDesk PCs to the simulator, for

example, to ask for names of the 3-D objects. Each port must have a unique number between 1025 and 65535.

Port #3 Lets you specify the UDP port number for the port that is used to send configuration responses from the simulator to the MotionDesk PCs, for example, to send the names of the 3-D objects. Each port must have a unique number between 1025 and 65535.

Network card Lets you choose the IP address of the MotionDesk PC.

Use specific simulator IP address Must be checked if if the MotionDesk PC has two or more network adapters.

Simulator IP Lets you specify the IP address of the simulator. The IP address is only required if the MotionDesk PC has two or more network adapters.

Stream buffer size Lets you specify the number of frames that are buffered by streaming. You should specify this number if the number of simulation frames is very high and buffering the whole simulation would take too long.

Frame buffer size Displays the number of frames that are buffered during the animation.

SCALEXIO page

This page is valid only for SCALEXIO system as the data source. It lets you choose a SCALEXIO Processing Unit or DS6001 Processor Board to provide the motion data for the animation.

Board Displays the information on the selected SCALEXIO Processing Unit or DS6001 Processor Board (name and IP address).

Select Board Lets you select a SCALEXIO Processing Unit or DS6001 Processor Board as the data source. If you click the button, a dialog opens that lists all the SCALEXIO Processing Units and DS6001 Processor Boards, found in the network. The list contains the board name, IP address and serial number.

To select a board, click its entry and then OK.

To update the list, click Update.

Unassign Board Lets you unassign the SCALEXIO Processing Unit or DS6001 Processor Board so that it is no longer used as a data source. If you do not unassign the board, no other user can access the simulation, unless the broadcast mode is used for the model.

Application Info Lets you select the real-time application to provide the motion data. All real-time applications are listed that run on the SCALEXIO Processing Unit or DS6001 Processor Board and include the MotionDesk services. When you select a real-time application, the values of Name, Path, Owner, and State are updated.

Property	Description
Name	Displays the name of the selected real-time application.
Path	Displays the path of the selected real-time application.
Owner	Displays the owner of the selected real-time application.

Property	Description
State	Displays the state of the selected real-time application.

VEOS page

This page is valid only if a VEOS is the data source. It lets you choose a VEOS to provide the motion data for the animation.

System Displays the information on the selected VEOS (name and IP address).

Select Local System Lets you select the MotionDesk PC as the data source. Click this button if VEOS and MotionDesk run on the same PC.

Select Network System Lets you select a VEOS as the data source which is connected via network to the MotionDesk PC. When you click this button, a dialog opens that lists all the VEOS found in the network. The list contains the system names, IP addresses, serial numbers, and information whether it is a local system. You can click the column headings to sort the list.

To select a platform, click its entry and then **OK**.

To update the list, click **Update**.

Unassign System Lets you unassign VEOS so that it is no longer used as a data source.

Application Info Lets you select the offline simulation application to provide the motion data. All offline simulation applications are listed that run on VEOS and include the MotionDesk services. When you select an offline simulation application, the values of **Name**, **Path**, **Owner**, and **State** are updated.

Name Displays the name of the selected offline simulation application.

Path Displays the path of the selected offline simulation application.

Owner Displays the owner of the selected offline simulation application.

State Displays the state of the selected offline simulation application.

Related topics

Basics

Data Source for Motion Data 15

HowTos

How to Register a Platform (ControlDesk Platform Management 🕮)
How to Select the Data Source 16

References

Simulation Data Sources 180

Video (Export)

Access

You can access this command via:

Ribbon	Home – Save As
Context menu of	None
Shortcut key	None
Icon	

Purpose

To generate a video of an animation.

Result

A video which can be played independently of MotionDesk is generated.

Dialog settings

This dialog lets you specify the parameters for a video export.

Start frame Lets you specify the first frame which is exported to a video.

End frame Lets you specify the last frame which is exported to a video.

Animation speed Lets you specify a factor to increase or decrease the speed of the animation.

Frame duration Lets you specify the duration for which a frame is displayed.

You have to specify the duration only if the MDF file was recorded with a MotionDesk version before MotionDesk 2.0. As of MotionDesk version 2.0, MotionDesk selects the frame duration automatically.

Frame rate Lets you specify the frame rate of the video. Normally, this is 25 frames per second.

You have to specify the frame rate only if the MDF file was recorded with a MotionDesk version before MotionDesk 2.0. As of MotionDesk version 2.0, MotionDesk selects the frame rate automatically.

Aspect ratio Indicates whether the ratio of width to height is set to 4:3. If you select the checkbox and change the width or height, the other value is changed in proportion.

Width Lets you specify the video width in pixel. The maximum width is the width of the 3-D View.

Height Lets you specify the video height in pixel. The maximum height is the height of the 3-D View.

Enable logo Lets you enable a logo that is displayed on the saved video.

The logo is saved in PNG file format in `<MotionDesk_InstallationPath>\MotionDesk\Assets\Pictures\VideoLogo.png`.

Codec Lets you choose the video codec format. All codec formats installed on your PC are listed.

Make sure to select a codec format that is available on the computers used for playing the video. As of MotionDesk version 3.9, you are recommended to use the Xvid MPEG-4 Codec.

You are recommended to install and use the newest version of the Microsoft Media Player.

File name Lets you specify a file name for the video. MotionDesk automatically suggests a file name. The name is made up of the experiment name and a number.

Generate Starts the video export.

Related topics

Basics

Generating a Video of an Animation..... 105

HowTos

How to Generate a Video..... 107

Simulation Data Sources

Access

You can access this command via:

Ribbon	Home – Platform
Context menu of	None
Shortcut key	None
Icon	

Purpose

To select a data source for animation.

Result

The selected data source provides the motion data for animation.

Description

A check mark indicates the active data source. You can also select the data source via Motion Player. The following data sources are available:

None No data source is selected, which means it is not possible to start an animation.

File: MDF: <Stream> Lets you select an MDF file as the data source. If checked, it displays the path and name of the currently selected MDF file. You can choose an MDF file on the **File** page of the **Customize Data Source** dialog. Refer to File page on page 174.

Network : UDP : Connection Lets you select a network adapter as the data source. You can choose a network adapter on the **Network** page of the **Customize Data Source** dialog. Refer to Network page on page 176.

VEOS : UDP : Connection Lets you select a VEOS as the data source. You can choose a VEOS on the **VEOS** page of the **Customize Data Source** dialog. Refer to VEOS page on page 178.

Hardware: Type : <Stream> Lets you select a real-time board that is connected via bus connection as the data source. If checked, it displays the real-time board where the application is running. You can choose a board on the **Hardware** page of the **Customize Data Source** dialog. Refer to Hardware page on page 174.

Hardware-Net: Type : <Stream> Lets you select a real-time board that is connected via network connection as the data source (MicroAutoBox II or modular hardware with slot CPU). If checked, it displays the real-time board where the application is running. You can choose a real-time board on the **Hardware-Net** page of the **Customize Data Source** dialog. Refer to Hardware-Net page on page 175.

SCALEXIO : UDP : Connection Lets you select a SCALEXIO system as the data source. You can choose a SCALEXIO Processing Unit or DS6001 Processor Board on the **SCALEXIO** page of the **Customize Data Source** dialog. Refer to SCALEXIO page on page 177.

DS1007 : UDP : Connection Lets you select a DS1007 as the data source. You can choose a DS1007 PPC Processor Board on the **DS1007** page of the **Customize Data Source** dialog. Refer to DS1007 page on page 174.

MLBX : UDP : Connection Lets you select MicroLabBox as the data source. You can choose MicroLabBox on the **MLBX** page of the **Customize Data Source** dialog. Refer to MLBX page on page 176.

ADAS : TCP : Connection Lets you select an ADAS simulator as the data source. You can choose ADAS on the **ADAS** page of the **Customize Data Source** dialog. Refer to ADAS page on page 173.

MABX III : UDP : Connection Lets you select MicroAutoBox III as the data source. You can choose MicroAutoBox III on the **MicroAutoBox III** page of the **Customize Data Source** dialog. Refer to MicroAutoBox III page on page 175.

Related topics

Basics

Data Source for Motion Data 15

HowTos

How to Select the Data Source 16

References

Customize Data Source 172

Go Online

Access

You can access this command via:

Ribbon	Home – Simulation
Context menu of	None
Shortcut key	**F5**
Icon	

Purpose

To start the transfer of motion data between the real-time processor and MotionDesk.

Result

The real-time application calculates the simulation data, which is transformed to motion data via the MotionDesk services, MotionDesk Blockset or the Model and Sensor Interface Blockset that is used for sensor simulation.

Description

Before you can start the online animation, you must prepare the selected data source, for example, download the real-time application. The Go Online command is disabled if no data source is selected, an MDF file is selected as data source, or if a real-time board is selected, but a real-time application is not running.

To start and stop the replay with an MDF file, use the Motion Player. Refer to Motion Player on page 184.

Related topics

Basics

Starting an Animation 50

Go Offline

Access

You can access this command via:

Ribbon	Home – Simulation
Context menu of	None
Shortcut key	**CTRL+F5**
Icon	

Purpose

To stop an online animation.

Result

MotionDesk stops reading the motion data from the selected data source. The online animation stops.

Related topics

HowTos

How to Store Motion Data 99

MDF

Access

You can access this command via:

Ribbon	Home - Save As
Context menu of	None
Shortcut key	**CTRL+Q**
Icon	
Others	In Motion Player

Purpose

To save the frame buffer as an MDF file.

Result

A standard Save As dialog opens to let you specify the file name and location.

Note

Do not use Asian characters, diacritic marks (such as ö or ó), and special characters (such as & : < > ' ") in the file path. MotionDesk does not support these characters.

Description

You can use the MDF file to replay an animation offline, for example, for presentation purposes on a system that is not connected to a real-time system.

Related topics

Basics

Data Source for Motion Data 15

HowTos

How to Store Motion Data 99

Motion Player

Access

You can access this dialog via:

Ribbon	View – Controlbars – Switch Controlbars
Context menu of	None
Shortcut key	None
Icon	None

Purpose

To control the replay of an animation.

Motion Player

Motion Player lets you replay an animation that has been saved as an MDF file or the current contents of the frame buffer.

Note

Some of the Motion Player commands are also available via the Home ribbon.

Data Source Lets you select a data source for animation. For more information, refer to Simulation Data Sources on page 180.

Browse Lets you select an MDF file, a real-time board, a network adapter, a SCALEXIO Processing Unit, or a VEOS as the data source. If an MDF file is selected in the Data Source list, a standard Open dialog appears, allowing you to select an MDF file. If another data source is selected, the Customize dialog appears, allowing you to edit the selected data source properties. For more information on the data source properties, refer to Customize Data Source on page 172.

Buttons The following buttons are available in Motion Player.

Button	Name	Description	Shortcut
	First Frame	Displays the first frame in the frame buffer.	**CTRL+1** (numeric keypad)
	Last Frame	Displays the last frame in the frame buffer.	**CTRL+3** (numeric keypad)
	Next Frame	Displays the next frame in the frame buffer.	**CTRL+9** (numeric keypad)
	Next Frames	Displays the next frames in sequence until the button is released.	None
	Previous Frame	Displays the previous frame in the frame buffer.	**CTRL+7** (numeric keypad)
	Previous Frames	Displays the previous frames in sequence until the button is released.	None
	Start Replay	Starts the animation replay from the current frame up to the last frame.	**CTRL+5** (numeric keypad)
	Stop Replay	Stops the animation replay.	**CTRL+4** (numeric keypad)
	Start Replay (Continuous Loop)	Starts the animation replay from the current frame and loops through the frames continuously until stopped manually.	**CTRL+6** (numeric keypad)
	Save As MDF File	Saves the frame buffer as an MDF file.	**CTRL+Q**

Player slider When you replay an animation, you can use the slider to move the animation forward and backward through the animation to a specific point. The values in the Time and Frame fields will change as you move the slider.

Note

You can move the slider and adjust the time and frame only when the Motion Player is stopped. The slider and the fields are disabled when the player is running the animation.

Time Displays the time value that belongs to the frame.

If the simulation model is implemented using the MotionDesk Blockset, you can select **Support recording synchronization** in the **MD_Communication** block. Then the displayed value is the time since the recording synchronization was triggered. Otherwise, the displayed value is the time since the simulation start.

If the simulation model is implemented using the Model and Sensor Interface Blockset, the displayed value is the time related to the maneuver start.

No time is shown for:

- MDF files earlier than MotionDesk 2.0
- Simulink simulations earlier than MotionDesk 2.0
- Multi-PC real-time applications earlier than MotionDesk 2.0

You can edit the time in this field to jump to a specific point in time in the animation.

Frame Specifies the number of the current frame. You can jump to a specific frame by entering the frame number in the box and pressing **Enter**.

Speed Lets you specify the speed of fast and slow motion. You can play the animation 2, 4, 8, 16, or 32 times faster or 1/2, 1/4, 1/8, 1/16, or 1/32 times slower.

Related topics

HowTos

How to Replay an Animation 103
How to Select the Data Source 16

References

MD_Communication (MotionDesk Calculating and Streaming Motion Data 🕮)
MDF 183

Motion Diagnostics

Access

You can access this dialog via:

Ribbon	View – Controlbars - Switch Controlbars
Context menu of	None

Shortcut key	None
Icon	None

Purpose

To display information about the translation and rotation of a movable object.

Result

During an animation, the **Motion Diagnostic** dialog displays the translation and rotation values of the currently selected movable object.

Description

The translation and rotation values are calculated on the real-time processor and transferred to MotionDesk in a transformation matrix. This information is helpful for debugging tasks during the setup phase of the kinematics transformations using the MotionDesk Blockset or the Model and Sensor Interface Blockset.

Dialog settings

Data Source Displays the currently selected data source. You can choose a different data source via the **Customize Data Source** command or via **Motion Player**. For more information, refer to Customize Data Source on page 172 and Motion Player on page 184, respectively.

Motion data Lets you choose which motion data object's translation and rotation values are displayed. The motion data names are held in the MDF file or defined in the model of the real-time application (namely in the **MD_Object** block. For more information, refer to MD_Object (MotionDesk Calculating and Streaming Motion Data 🕮).

Rotation Displays the values of the rotation matrix for the currently selected motion data. The values are updated only when an animation is running. For more information on the rotation matrix, refer to Homogeneous Transformation (MotionDesk Calculating and Streaming Motion Data 🕮) and the topics following it.

Translation Displays the values of the translation vector for the currently selected motion data. The values are updated only when an animation is running. For more information on the translation vector, refer to Homogeneous Transformation (MotionDesk Calculating and Streaming Motion Data 🕮).

PHI Displays the first rotation angle of a set of Euler or roll-pitch-yaw angles.

THETA Displays the second rotation angle of a set of Euler or roll-pitch-yaw angles.

PSI Displays the third rotation angle of a set of Euler or roll-pitch-yaw angles.

Mode Indicates which mode the angles are displayed in. You can specify Euler or Roll, Pitch, Yaw. For more information on these types of angles, refer to Euler Angles (MotionDesk Calculating and Streaming Motion Data 🕮) and Cardan

Roll, Pitch, and Yaw Angles (MotionDesk Calculating and Streaming Motion Data 🕮).

Format Indicates which units the angles are displayed in. You can specify Degrees or Radians.

Related topics

Basics

Simulation Data Mathematical Principles (MotionDesk Calculating and Streaming Motion Data 🕮)

References

MD_Object (MotionDesk Calculating and Streaming Motion Data 🕮)

Scene Manager Properties

Introduction

The Scene Manager provides various properties to specify the characteristics for the scene animation.

Where to go from here

Information in this section

Atmospherics Properties

Purpose

To specify the atmosphere for the scene.

Description

You can use sets of predefined values or create your own by editing the values in the Custom mode. If you select preset atmospheres, the property values are read-only.

General properties

Name Displays the name of the atmospherics.

Mode Lets you select a preset of atmosphere settings or the mode to specify your own settings.

Mode	Description
Custom	Lets you specify your own atmosphere settings. You can modify the atmosphere properties only if you select this mode.

Mode	Description
Foggy	Sets the property values for a foggy atmosphere.
Hazy	Sets the property values for a hazy atmosphere.
Night	Sets the property values for a night atmosphere.
Rainy	Sets the property values for a rainy atmosphere.
Snowy	Sets the property values for a snowy atmosphere.
Sunny	Sets the property values for a sunny atmosphere.
Sunset	Sets the property values for a sunset atmosphere.

Apply preset Lets you select an atmospherics preset.

Tone mapping Let you enable tone mapping for the atmospheric settings to improve the range of light intensities in a scene and retain the details in the image.

Sunlight

This section lets you specify the sunlight settings for your atmospheric.

Azimuth angle Lets you specify the azimuth angle of the sunlight in degrees. Values range from 0° degrees to 360°. You can edit the value by entering a value or by selecting it by using the arrows of the cell.

Color Lets you select the color of the sunlight. To edit the color value open the standard color select dialog by clicking the arrow in the cell. You can select colors from a basic color palette or define custom colors.

Elevation angle Lets you specify the elevation angle of the sunlight in degrees. Values range from 0° to 90°. You can edit the value by entering a value or by selecting it by using the arrows of the cell.

Intensity Lets you specify the intensity of the sunlight. The sunlight is generated by a point light which is far away. Its rays are parallel in the scene. Values range from 0 to 1. You can edit the value by entering a value or by selecting it by using the arrows of the cell. The default intensity of sunlight is 1.

Fog

This section lets you specify the fog settings for your atmospheric.

Color Lets you specify the color of the sunlight. To edit the color value open the standard color select dialog by clicking the arrow of the value cell. You can select colors from a basic color palette or define custom colors.

Density Lets you select the course of the fog density from the headlamp to the visibility range. You can choose between Exponential or ExponentialSquared. You can select the density value by using the arrow of the cell.

Enabled Lets you enable or disable the appearance of fog in your atmospherics. If you want fog to appear in your atmospheric, you have to activate the checkbox.

Range Lets you specify the visibility range of fog. All objects behind the visibility range are not shown. You can edit the value by entering a value or by selecting it by using the arrows of the cell.

Rain

This section lets you specify the settings for your rainy atmospheric.

Color Lets you specify the color of the rain particles. To edit the color value, open the standard color select dialog by clicking the arrow of the value cell. You can select colors from a basic color palette or define custom colors.

Enabled Lets you enable rain in your atmospherics.

Falling angle Lets you specify the falling angle of the rain particles. If the angle is 90%, the particles are falling vertically and the heading angle has no effect. If the angle is 0%, the particles are falling horizontally.

Heading angle Lets you specify the heading angle of the rain particles.

Intensity Lets you specify the intensity of the rain.

Particle size Lets you specify the size of the rain particles.

Speed Lets you specify the speed of the rain particles.

Snow

This section lets you specify the settings for your atmospheric with falling snow.

Color Lets you specify the color of the snow particles. To edit the color value, open the standard color select dialog by clicking the arrow of the value cell. You can select colors from a basic color palette or define custom colors.

Enabled Lets you enable snow in your atmospherics.

Falling angle Lets you specify the falling angle of the snow particles. If the angle is 90%, the particles are falling vertically and the heading angle has no effect. If the angle is 0%, the particles are falling horizontally.

Heading angle Lets you specify the heading angle of the snow particles.

Intensity Lets you specify the intensity of the falling snow.

Particle size Lets you specify the size of the snow particles.

Speed Lets you specify the speed of the snow particles.

Shadow

This section lets you specify the settings for shadow in the scene.

Distance Lets you specify the distance that the of the shadow is rendered.

Enabled Lets you enable shadow globally. When this option is disabled, no 3-D object in the scene casts a shadow. When this option is enabled, objects cast shadows only if their Cast shadow property is enabled.

Transparency Lets you specify the transparency of the shadows. This contributes to the shading of the objects in the scene.

Related topics

HowTos

How to Specify Own Atmospheric Settings..... 131

Instrument Properties

Purpose

To view or edit the properties of an instrument or an instrument panel.

Description

You can assign an instrument to a movable object or select a static position. The default for a new instrument is *static*. A static instrument is fixed at the bottom left corner of a window in the 3-D View. A dynamic instrument is fixed to the movable object that you selected. If you change from a static instrument to a dynamic one, the position is automatically set to the values *X = 0* and *Y = 0*.

To be able to use instruments in the scene, the real-time model must be prepared. Refer to Using Instruments in the Scene on page 31.

Depending on the instrument type, different instrument properties are available.

Background properties

Background color Lets you set the background color of an instrument.

Enabled Lets you enable the background for an instrument. The option can be enabled only for instruments with a numeric display.

Has picture Lets you specify whether you want a picture in the background.

Picture path Lets you select a file that is used as the background image.

Style Lets you select the style of the picture. For example, you can stretch the image so that it fits the instrument.

Child instrument properties

The properties are available only for instruments belonging to an instrument panel.

X-offset to Frame [pixel] Lets you specify the x-offset of a child instrument to its parent in pixels.

Y-offset to Frame [pixel] Lets you specify the y-offset of a child instrument to its parent in pixels.

Is primary child Displays whether the selected child instrument is the primary child instrument of an Instrument Panel. To make a child instrument as primary child, use the Make Primary command of the Scene Navigator's context menu.

For information on how to configure a primary child instrument, refer to How to Configure the Title Bar of a Minimized Instrument Panel on page 40.

Collapsed mode properties

Background color Lets you set the background color of an instrument that is minimized.

Background enabled Lets you enable the background for a numeric display. The option is enabled only for the instruments of numeric type and instruments with an additional numeric display.

Is collapsible Lets you specify whether the Instrument Panel can be minimized.

Is collapsed Lets you specify whether the Instrument Panel is minimized.

Tip

You can minimize or maximize an Instrument Panel via mouse operations in the 3-D View:

- To minimize an Instrument Panel, press the **Shift** key and click the minus sign.
- To maximize an Instrument Panel, press the **Shift** key and click the plus sign.

Text font Lets you specify the font for the text that is displayed when the Instrument Panel is minimized.

Note

MotionDesk's instruments do not support script fonts, such as the *Vladimir Script* font. You can identify script fonts by their name.

Title text Lets you specify the text that is displayed in the title bar when the Instrument Panel is minimized. The title text can display the values of the signal assigned to primary child instrument. You can use macros to displays the values, refer to the following table:

Macro	Description
`{%SIGNALNAME%}`	Name of the signal as specified in the simulation model.[1)]
`{%DATATYPE%}`	Data type of the signal as specified in the simulation model.[1)]
`{%UNIT%}`	Unit of the signal as specified in the simulation model.[1)]
`{<printf format string>%VALUE%}`	Value of the signal. You can optionally specify a printf format string to format the output of the value. For

Macro	Description
	example, to output an integer value with at least 5 digits, enter: `{%<%5d>VALUE%}`.

[1] Refer to MD_Instrumentation (MotionDesk Calculating and Streaming Motion Data 🕮).

For details on configuring the title text, refer to How to Configure the Title Bar of a Minimized Instrument Panel on page 40.

Title text color Lets you specify the color of the text that is displayed when the Instrument Panel is minimized.

Color

Background color Lets you select the background for a numeric display. For a new instrument, the checkbox is cleared by default. The options are enabled only for the Numeric type and instruments with an additional numeric display. The default for a new instrument is white. Click the rectangle in the edit field to open a standard color selection dialog.

Background enabled Lets you enable or disable the background for a numeric display. The options are enabled only for the Numeric type and instruments with an additional numeric display.

Instrument Lets you select the color of the instrument. Click the rectangle in the edit field to open a standard color selection dialog.

The default for a new instrument is gray. The color selected here is used for the following purposes:

Instrument	Display
Numeric	Caption, unit
Gauge	Scale
Bar	Scale
Multistate LED	Frame

Instrument properties

Attached to Lets you assign an instrument to a movable object or select a static position. The default for a new instrument is static. A static instrument is fixed at the bottom left corner of a window in the 3-D View. A dynamic instrument is fixed to the movable object that you selected. If you change from a static instrument to a dynamic one, the position is automatically set to the values *X = 0* and *Y = 0*.

Font size Lets you specify the font size of the numbers in pixel in the range 4 … 100. The default value is 14.

Height Lets you define the height of an instrument in the 3-D View as a percentage of the associated window. You cannot enter a height for Gauge or round LED instruments.

Is vertical Lets you select whether a Bar instrument is horizontal or vertical.

Is visible Lets you specify whether the selected instrument is visible in the scene.

Name Lets you enter a unique name for the instrument. Each new instrument is automatically given a default name, which you can change. The name is displayed in the Scene Navigator to identify the instrument.

Observer Lets you select the observer you want to link the instrument to. The instruments are dependent on the view shown in the 3-D View. The appropriate observers must be selected there. You can select several observers. In the Scene Navigator, the instruments are listed below the appropriate observers. The default observer for a new instrument is the observer belonging to the active window in the 3-D View.

An instrument must always be linked to observers. If you delete all links to observers of an instrument, MotionDesk automatically links the instrument to the Free1 observer.

Tip

You can also assign the instruments to the observers via drag & drop in the Scene Navigator: Drag the instrument from the Instruments folder to the observer.

Range Lets you specify the value range for a simulation variable. The range also determines the colors that the instrument displays.

Signal data Lets you select a simulation variable to display its value on the instrument. Up to 300 simulation variables can be transferred from the Simulink model. To be transferred to MotionDesk, a simulation variable must be connected to the MD_Instrumentation block in the simulation model. Refer to How to Prepare the Real-Time Model for Using Instruments or State Objects (MotionDesk Calculating and Streaming Motion Data 🕮). A new instrument is not connected by default. If an instrument is unconnected, its icon in the Scene Navigator has an question mark.

Type Lets you select a type for an instrument. The available types are Numeric, Gauge, Bar, Multistate LED, Multistate Picture, and Multistate Text.

The default type for a new instrument is Numeric. If you change the type, various input options are switched on or off, and all the relevant data is transferred from the original instrument to the new one.

Unit Displays the unit of the simulation variable.

Width Lets you define the width of an instrument in the 3-D View as a percentage of the associated window.

LED

The properties are available only for Multistate LED instruments.

Form Lets you specify the form for Multistate LED instruments. The selections you make determine what these instruments look like. A Multistate LED instrument can be rectangular or round. A new instrument is round by default.

Color/frame color Lets you specify the color or frame color of the instrument.

LED states

The properties are available only for Multistate LED instruments.

States Lets you specify the states and the corresponding colors for a Multistate LED instrument. When you click the Browse button, the States dialog opens for you to specify the settings. Refer to States Dialog on page 202.

Picture

The properties are available only for Multistate Picture instruments.

Alignment Lets you specify the alignment of the picture if the Style property is set to 'Single'.

Style Lets you use the Width and Height properties to select how the picture in the area is specified.

Style	Description
Stretched	The picture is displayed once in the area. It is stretched or compressed over the area to fill the complete area.
Single	The picture is displayed once inside the area in its original resolution. If the picture is wider or higher then the area, it is truncated. The alignment is specified by the Alignment property.
Tiled	The picture can be displayed several times in the area in its original resolution. If the picture is smaller or lower then the area, the picture is repeatedly displayed. If the picture is wider or higher then the area, it is truncated.

Picture states

The properties are available only for Multistate Picture instruments.

States Lets you specify the states and the corresponding pictures for a Multistate Picture instrument. When you click the Browse button, the States dialog opens for you to specify the settings. Refer to States Dialog on page 202.

Position and size

Autopositioning enabled Lets you enable the automatic repositioning of overlapping instruments that are attached to a movable object.

Positioning Displays the current values for the x-offset and y-offset.

Positioning mode Lets you select whether the offset values for positioning are specified as a percentage of the active window or in pixel.

When you switch the mode, MotionDesk converts the values of the x-offset and y-offset. If a new value in percentage is above 100%, it is set to the default value of 40%.

X-Offset Lets you specify the x-value of the position as a percentage of the active window of the 3-D View or as an absolute value in pixel. The origin of a static instrument is defined absolute to the bottom left corner. The origin of a dynamic instrument is relative to the assigned movable object. The default for a new instrument is 40%.

Y-Offset Lets you specify the y-value of the position as a percentage of the active window of the 3-D View or as an absolute value in pixel. The origin of a static instrument is defined absolute to the bottom left corner. The origin of a dynamic instrument is relative to the assigned movable object. The default for a new instrument is 40%.

Numeric

The properties of the Numeric category can be specified for Gauge, Bar, Multistate Picture and Multistate LED instruments.

Alignment Lets you select one of 9 positions for an additional numeric display for a Bar, Multistate Picture, or Multistate LED instrument. The options are enabled only for instruments with an additional numeric display.

Background color Lets you set the background color of the numeric instrument.

Background enabled Lets you enable the background for a numeric instrument. The options are enabled only for instruments with a numeric display.

Decimal Lets you specify the number of digits after the decimal point (decimal places) in a numeric display. You can use this option only for the Numeric instrument type and for Gauge, Bar, Multistate Picture, and Multistate LED instruments that have an additional numeric display. The default for a new instrument is 1. Only the number of digits specified here is displayed. The value is rounded.

The following table shows the display for different decimal places numbers of the simulation variable = 123.45608473.

Decimal Places Number	Display
0	123
1	123.5
2	123.46
3	123.456
4	123.4561

Enabled Lets you add a numeric display to a Gauge, Bar, Multistate Picture, or Multistate LED instrument. The display is positioned relative to the actual

instrument and inherits its properties. For a new instrument, the checkbox is cleared by default. You cannot use this option for the Numeric instrument type.

Font size Lets you specify the font size of the numeric display in pixel in the range 4 … 100. The default value is 14.

Numerical value color Lets you specify the color of the numerical value of a Multistate Picture instrument.

Predecimal Lets you specify the number of digits before the decimal point (integer part) in a numeric display. If the specified number is greater than the number of digits to be displayed, placeholders are inserted automatically. All the digits of the value's integer part are displayed even if they are more than the specified number.

You can use this option only for the Numeric instrument type and for Gauge, Bar, Multistate Picture, and Multistate LED instruments that have an additional numeric display. The default for a new instrument is 6.

Example: The following table shows the display for different predecimal values of the simulation variable = 123.456.

Predecimal Digits Number	Display
0	123.456
1	123.456
2	123.456
3	123.456
4	123.456
5	123.456
6	123.456

Range

The properties of the Range category can be specified for Numeric, Gauge, and Bar instruments.

Max Lets you specify a upper bound for the simulation variable. The default is 1000.

Min Lets you specify a lower bound of the simulation variable. The default is 0.

Over range color Lets you select the color for values that are higher than the Max value. The default for a new instrument is red. Click on the rectangle in the edit field to open a standard color selection dialog.

Under range color Lets you select the color for values that are lower than the Min value. The default for a new instrument is yellow. Click on the rectangle in the edit field to open a standard color selection dialog.

Within range color Lets you select the color for values that are inside the range between Min and Max. The default for a new instrument is green. Click on the rectangle in the edit field to open a standard color selection dialog.

Scale

The properties are available only for Gauge and Bar instruments.

Major delta Lets you specify the distance between major tics. The distance between 2 major tics can be in the range 0 ≤ value ≤ (**Max value** – **Min value**). If the value is 0, no scale is displayed. The default for a new instrument is 20.

Major start Lets you specify the position of the first major tic on the scale. The default for a new instrument is 0. The value must be within the value range of the scale (**Max value** – **Min value**).

Max value Lets you define the end of the scale. The defaults for a new instrument is 240. If the maximum value equals the minimum value, the scale is automatically removed.

Min value Lets you define the start of the scale. The defaults for a new instrument is 0. If the maximum value equals the minimum value, the scale is automatically removed.

Minor count Lets you specify the number of minor tics between 2 major tics. The default for a new instrument is 3. If the value is 0, no minor tics are displayed. The scale must be active and the major delta distance must be greater than 0.

Text

The properties are available only for Multistate Text instruments.

Alignment Lets you select the alignment of the string.

Font Lets you select the font, font style, and size of the string.

Note

MotionDesk's instruments do not support script fonts, such as the *Vladimir Script* font. You can identify script fonts by their name.

Text states

The properties are available only for Multistate Text instruments.

States Lets you specify the states and the corresponding strings for a Multistate Text instrument. When you click the Browse button, the **States** dialog opens for you to specify the settings. Refer to States Dialog on page 202.

Related topics

Basics

Basics on Using Instrument Panels in the Scene....36
Basics on Using Instruments in the Scene....31

HowTos

How to Use Instrument Panels in the Scene....37

How to Use Instruments in the Scene 33

References

Create (Instrument) 144
Make Primary 158
MD_Instrumentation (MotionDesk Calculating and Streaming Motion Data 🕮)

Observer Properties

Purpose

To view or edit the properties of an observer.

Properties

Attached to Lets you select a movable object to which the observer is attached to. The property is available only if the observer's behavior depends on a movable object.

The property is not displayed for the default free observers **Free1** and **Free2**.

Behavior Lets you select the behavior of the observer. The following behaviors are available:

The property is not displayed for the default free observers **Free1** and **Free2**.

Behavior	Description
Free	Observer can be moved freely in the virtual world.
Fixed inertial	Observer is fixed to the virtual world.
Fixed to object	Observer is fixed to a movable object.
Follow driver	Observer follows the attached movable object. The view is from the driver's seat.
Follow driver smooth	Observer follows the attached movable object smoothly. The view is from the driver's seat.
Follow rotation	Observer follows the rotation of the attached movable object.
Follow translation	Observer follows the translation of the attached movable object.
Follow X-Translation	Observer follows the x translation of the attached movable object.
Follow Y-Translation	Observer follows the y translation of the attached movable object.

Note

When you select the Free observer behavior, only the position and orientation offset properties can be edited.
For Fixed and Follow observer behaviors, only the position and orientation properties can be edited.

Name Lets you specify the name of the selected observer.

Locked (fixed observers only) Lets you lock and unlock the position and orientation of the observers. When locked, the observer view returns to the original position when the mouse button is released after moving around the 3-D View. If unlocked, the position and orientation offsets are adjusted and remain at the new position when you release the mouse button.
The property is not displayed for the default free observers Free1 and Free2.

Position offset (fixed observers only)

The following position offset properties are available only for fixed observers.

X-Offset Lets you specify the offset in x-direction in meters. This is the distance of the observer to the attached movable object or the virtual world (depending on the observer's behavior).

Y-Offset Lets you specify the offset in y-direction in meters. This is the distance of the observer to the attached movable object or the virtual world (depending on the observer's behavior).

Z-Offset Lets you specify the offset in z-direction in meters. This is the distance of the observer to the attached movable object or the virtual world (depending on the observer's behavior).

Orientation offset (fixed observers only)

The following orientation offset properties are available only for fixed observers.

X-Rotation Lets you specify the rotation around its x-axis in degrees within the range -360° ... 360°.

Y-Rotation Lets you specify the rotation around its y-axis in degrees within the range -360° ... 360°.

Z-Rotation Lets you specify the rotation around its z-axis in degrees within the range -360° ... 360°.
The property is gray for the Follow rotation behavior.

Position (free observers only)

The following position offset properties are available only for free observers.

X-Position Lets you specify the x-position of the free observer in meters.

Y-Position Lets you specify the y-position of the free observer in meters.

Z-Position Lets you specify the z-position of the free observer in meters.

Orientation (free observers only)

The following orientation offset properties are available only for **fixed** observers.

X-Orientation Lets you specify the orientation of the free observer around its x-axis in degrees. The value must be in the range -360° ... 360°.

Y-Orientation Lets you specify the orientation of the free observer around its y-axis in degrees. The value must be in the range -360° ... 360°.

Z-Orientation Lets you specify the orientation of the free observer around its z-axis in degrees. The value must be in the range -360° ... 360°.

Related topics

HowTos

How to Create and Configure a Custom Observer 82
How to Create Default Observers 81

States Dialog

Access

You can open the dialog via the Browse button of the **States** property of a Multistate LED, Multistate Picture, or Multistate Text instrument.

Purpose

To specify the states and the associated colors of Multistate LED, Multistate Picture, and Multistate Text instruments.

Description

A Multistate instrument displays the value of a variable as a state. In the **States** dialog, you can specify several states with ranges for the value and corresponding appearances (colors, pictures, or strings).

When a value is in the range of several states, the appearance of the state that is first in the list is used.

When the value is outside of all the specified ranges of the states, the instrument has the appearance of the default state.

When a Multistate instrument is created, it has four states by default:

- **Default state**
- **Under range**
- **Within range**
- **Over range**

You cannot delete the **Default state** but modify its name or appearance. The other three states can be deleted or modified.

Dialog settings

Buttons Lets you add and remove states and specify the order.

Button	Purpose
Add	To add a new state.
	To move the selected state upwards in the list.
	To move the selected state downwards in the list.
	To delete the state from the list.

Color (only Multistate LED instruments) Opens a dialog where you can select a color or enter values to define one. To close the dialog, double-click a color or press **Enter**.

Include maximum Lets you specify whether to include the maximum limit in the range.

Include minimum Lets you specify whether to include the minimum limit in the range.

Maximum Lets you enter the maximum limit of the range for the selected state.

Minimum Lets you enter the minimum limit of the range for the selected state.

Name Lets you enter the name of the state.

Picture path (only Multistate Picture instruments) Opens a dialog where you can select the file containing the picture. MotionDesk supports the following file types: BMP, JPG, PNG, and TIFF format.

Text (only Multistate Text instruments) Lets you enter the string to be displayed. It is possible to use macros in the string. MotionDesk replaces the macros by appropriate value during run time. The following table displays the macros which can be used.

Macro	Description
`{%SIGNALNAME%}`	Name of the signal as specified in the simulation model.[1)]
`{%DATATYPE%}`	Data type of the signal as specified in the simulation model.[1)]
`{%UNIT%}`	Unit of the signal as specified in the simulation model.[1)]
`{<printf format string>%VALUE%}`	Value of the signal. You can optionally specify a printf format string to format the output of the value. For

Macro	Description
	example, to output an integer value with at least 5 digits, enter: `{%<%5d>VALUE%}`.

[1] Refer to MD_Instrumentation (MotionDesk Calculating and Streaming Motion Data 🕮).

Text color (only Multistate Text instruments) Opens a dialog where you can select a color or enter values to define one. To close the dialog, double-click a color or press **Enter**.

Related topics

HowTos

How to Use Instruments in the Scene..... 33

References

Instrument Properties..... 192
MD_Instrumentation (MotionDesk Calculating and Streaming Motion Data 🕮)

Track Properties

Purpose

To view or edit the properties of a track.

Properties

Auto update Lets you select whether the track copies the latest frames of a simulation automatically after simulation end. An activated checkbox indicates, that the auto-function is enabled. Only one track in an experiment can use auto update.

Clear Buffer Clears the track buffer.

Data source Lets you specify the source for the track's data.

Fetch Simulation Data Copies data from the simulation buffer to the track buffer.

MDF File Lets you select an MDF file as data source of the track. You can use the Browse button to select the file.

Name Lets you specify the name of the track.

Play in loop Lets you specify whether the track is played in a loop. An activated checkbox indicates that the track is played in a loop.

Save as MDF file Saves the current content of the simulation buffer to an MDF file.

Time offset (sec) Lets you specify the delay of the track's start time in seconds.

Related topics

HowTos

How to Configure Track Properties..... 117

3-D View Properties

Purpose

To view or edit the properties of a view that is displayed in the 3-D View.

Properties

Far clipping plane Lets you specify the distance of the far clipping plane.

Near clipping plane Lets you specify the distance of the near clipping plane.

Name Lets you specify the name of the view.

Observer Lets you select the current observer used in this view.

Horizontal/vertical view angle Lets you specify the horizontal/vertical view angle of this view. The other view angle (vertical/horizontal) is calculated according to the view's aspect ratio.

Show material preview colors Let you specify if you want the MotionDesk observer view to display the material preview colors.

Note

You can select Show material preview colors only if Enable pseudo color is not selected.

Tone mapping Let you enable tone mapping that is shown in the scene in the 3-D View to improve the range of light intensities in a scene and retain the details in the image.

Exposure Let you specify the amount of light exposure to be shown in the scene in the 3-D View. Increasing the exposure brightens the scene. You can specify a range between 0 and 100. The default value is 1.

Bird's-eye view

Enable Lets you enable the bird's-eye view for the selected view. This is a two-dimensional view above the scene that is shown in the 3-D View.

You can roll the mouse wheel button to increase and decrease the area of the scene displayed in the bird's-eye view. This has the effect of zooming in and out above the scene. You can also specify the height and width in the properties of the scene that is visible for the bird's-eye view.

If an observer is fixed to a movable object in the selected view that is assigned to motion data, the bird's-eye view follows the object through the scene.

Height Lets you specify the height in meters of the scene along the latitude that is displayed by the bird's-eye view. If you change the height, the width of the scene also changes.

Width Lets you specify the width in meters of the scene along the longitude that is displayed by the bird's-eye view. If you change the width, the height of the scene also changes.

Pseudo color

Enable pseudo color Let you specify if you want the 3-D view to display the pseudo colors.

Note

You can only select Enable pseudo color and configure the pseudo color ranges if Show material preview colors is not selected.

Pseudo color range settings Opens a dialog for you to specify the pseudo color rendering range for the pseudo coloring displayed in the scene.

You can set move the sliders to the adjust the primary color percentage values or set the percentages manually in the color fields.

The Color scale changes to show the graphical color representation of the color range that you specify.

Related topics

Basics

Viewpoints, Clipping Plane (MotionDesk Basics 🕮)

HowTos

How to Configure Scene Pseudo Coloring in MotionDesk....66
How to Configure the 3-D Views....55
How to Configure Tone Mapping and Exposure....70
How to Display the Material ID Preview in MotionDesk....60
How to Use the Bird's-eye View....58
Working with Pseudo Coloring in the MotionDesk 3-D View....62

Dialog and Pages

Introduction

The Scene Manager provides various dialog pages to specify the settings for the scene animation.

Where to go from here

Information in this section

Graphics Capabilities Options Page

Access

This page is part of the **MotionDesk Options** dialog.

Purpose

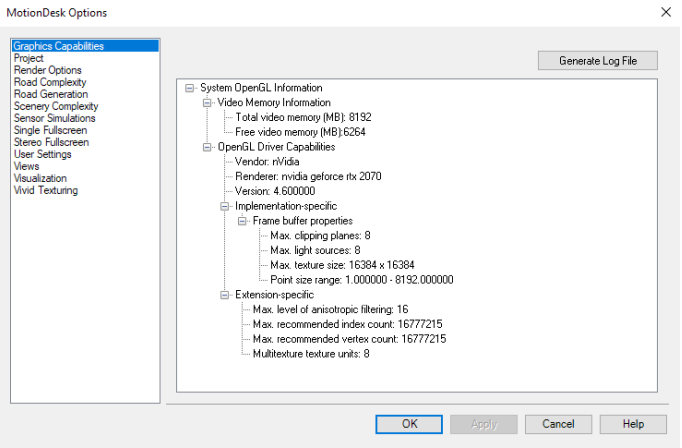
To display information on the OpenGL capabilities of the graphics adapter.

Note

The graphics adapter must be equipped with OpenGL hardware acceleration and with a minimum of 2 GB texture memory. Otherwise you may not be able to create or load an experiment properly.

Descripton

You can see the graphics capabilities for MotionDesk.



Dialog settings

Generate Log File Stores the information on the graphics adapter into a log file.

Related topics

References

MotionDesk Options (MotionDesk Basics 🕮)

Render Options Page

Access

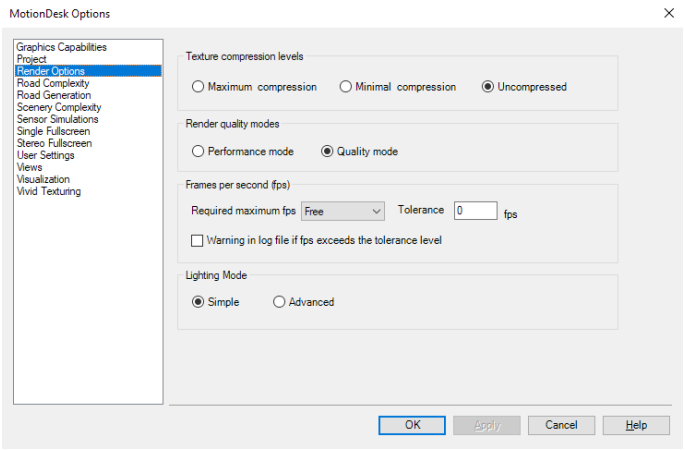
This page is part of the MotionDesk Options dialog.

Purpose

To specify the settings for texture compression.

Description

MotionDesk can compress the textures of 3-D objects to save graphics memory. You can use this option if your experiment has a large scene. Because the objects are compressed while the MotionDesk project is loaded, the loading time is increased.



Dialog settings

Texture compression levels Lets you select which kind of textures are compressed.

Option	Description
Maximum compression	All textures are compressed.
Minimal compression	Only textures for color rendering are compressed.
Uncompressed	No textures are compressed.

Render quality modes Lets you select the renderer mode (setting of antialiasing and texture filtering).

Mode	Description
Performance mode	To deactivate antialiasing and texture filtering.
Quality mode	To activate antialiasing and texture filtering.

For details on the renderer mode, refer to How to Activate Anti-aliasing and Texture Filtering on page 125.

Required minimum fps Lets you select a frame rate that must be reached at least for the animation in MotionDesk. If you select 'free', MotionDesk captures all processor time which is not needed by other programs. The total CPU usage is always 100%. If you select a value, this is the maximum frame rate.

Tolerance Lets you specify a tolerance value for the required minimum frames per second.

Warning in log file if fps exceeds the tolerance level Lets you select whether a warning message is displayed in the log file when the actual frame rate exceeds the required frame rate and tolerance level.

Lighting mode Lets you select the lighting mode:

Mode	Description
Simple	To use simple lighting in the scene.
Advanced	To use advanced lighting in the scene. In this mode, you can add additional light objects to static and movable objects that provide the scene with additional illumination.

In the advanced lighting mode, the illumination of the scene is higher than in the simple lighting mode. The advanced lighting mode calculates the illumination of a scene using sunlight and reflections of light sources, i.e., even objects are illuminated that are in the shade. Therefore, scenes in the advanced lighting mode look brighter than in the simple lighting mode. To adjust this, you can reduce the intensity of the sunlight in the Atmospherics properties. Refer to Atmospherics Properties on page 189.

Related topics

Basics

Basics of the Advanced Lighting Mode (MotionDesk Scene Creation 🕮)

HowTos

How to Activate Anti-aliasing and Texture Filtering 125
How to Limit the Frame Rate 53
How to Set the Rendering and Lighting Levels 52

References

MotionDesk Options (MotionDesk Basics 🕮)
Road Generation Options Page (MotionDesk Scene Creation 🕮)

Single Fullscreen Options Page

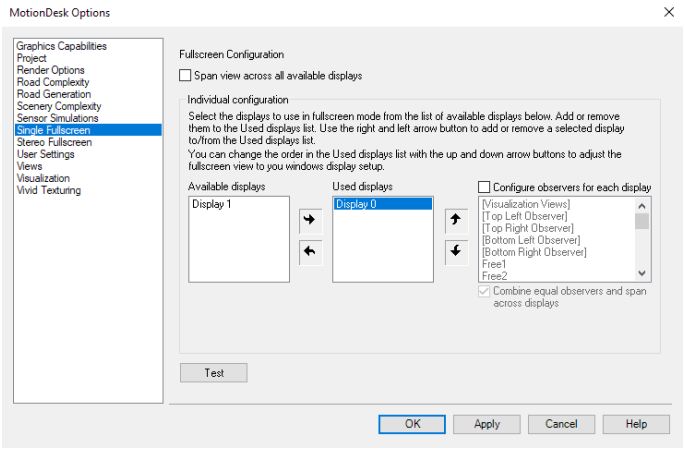
Access This page is part of the MotionDesk Options dialog.

Purpose To configure the full screen mode.

Description If the MotionDesk PC is connected to several displays, you can use all or some of them for visualization. MotionDesk can spread one view across all the available displays, or you can configure the displays yourself.

The observers used in the views are assigned to the displays by default as follows:

Display	View
Display 0	Top left window
Display 1	Top right window
Display 2	Bottom left window
Display 3	Bottom right window



Dialog settings

Span view across all available displays Lets you enable all the available displays for the full screen mode. Select this option if more than one display is connected to the MotionDesk PC and you want to use all of them for visualization. Otherwise, clear this option to select the displays and configure the observers.

Available displays Lists all the displays that are available for the MotionDesk PC.

To select a display for MotionDesk, select it in the table and click .

Used displays Lists all the displays that are used by MotionDesk.

To unselect a display, select it in the table and click .

To change the order of the displays, select a display in the table and click and .

Configure observers for each display Lets you select an observer for each display used. You can select an observer by name or an observer which is already selected in a view, see the following table.

Item	Description
[Visualization Views]	In the full screen mode, the selected display shows four views. The views use the same observers as in normal mode.

Item	Description
[Top Left Observer]	In the full screen mode, the selected display uses the observer of the top left view.
[Top Right Observer]	In the full screen mode, the selected display uses the observer of the top right view.
[Bottom Left Observer]	In the full screen mode, the selected display uses the observer of the bottom left view.
[Bottom Right Observer]	In the full screen mode, the selected display uses the observer of the bottom right view.

Combine equal observers and span across displays Lets you combine views that have the same observer across the displays.

Test Tests the configuration. Press **Esc** to finish the test.

Related topics

HowTos

How to Configure Full Screen Mode with Multiple Displays..... 73

References

MotionDesk Options (MotionDesk Basics 🕮)
Single Full Screen..... 165

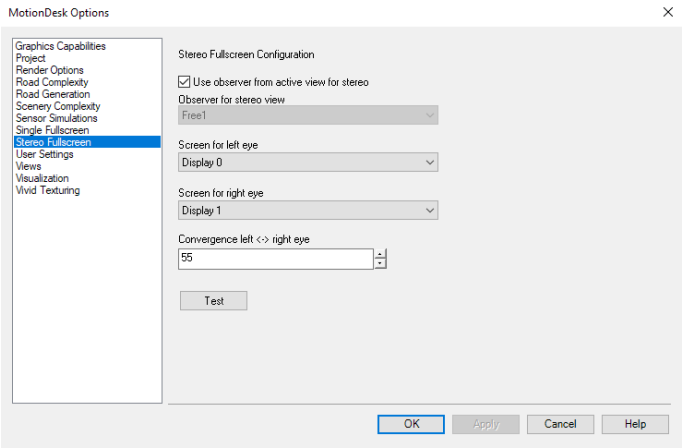
Stereo Fullscreen Options Page

Access This page is part of the MotionDesk Options dialog.

Purpose To configure the stereo full screen mode.

Description

Select full screen configuration for the observer view and split the view across two screens.



Dialog settings

Use observer from active view for stereo Lets you enable the observer of the active view. If the option is cleared, you can select an observer in the field below.

Observer for stereo view Lets you select an observer to use for the stereo view.

Screen for left eye Lets you select a display to display the view for the left eye.

Screen for right eye Lets you select a display to display the view for the right eye.

Convergence left <-> right eye Lets you specify the convergence between the left and right eyes, for example, the distance between the eyes or cameras.

Test Tests the settings. Press **Esc** to finish the test.

Related topics

HowTos

How to Configure a Stereo Display..... 76

References

MotionDesk Options (MotionDesk Basics 🕮)
Stereo Full Screen..... 166

Visualization Options Page

Access

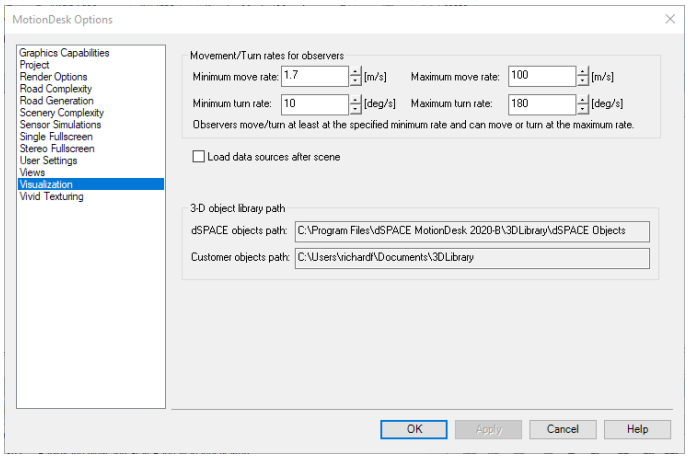
This page is part of the MotionDesk Options dialog.

Purpose

To alter the properties in MotionDesk related to visualization.

Description

To configure the visualization properties for MotionDesk, for example, the movement and turn rates for the observers in the zone navigation mode and the location of the 3-D object library.



Dialog settings

Minimum move rate Lets you specify the minimum rate in m/s for moving observers.

Maximum move rate Lets you specify the maximum rate in m/s for moving observers.

Minimum turn rate Lets you specify the minimum rate in deg/s for turning observers.

Maximum turn rate Lets you specify the maximum rate in deg/s for turning observers.

Load data sources after scene Lets you specify the sequence of loading the MDF file and the scene. If the option is enabled, the MDF file is loaded after the scene. Otherwise, the MDF file is loaded first.

3-D object library path Displays the paths of the 3-D object libraries. MotionDesk uses these paths to search for the 3-D objects for creating a scene.

- dSPACE objects path is the path to the dSPACE objects library, which is read-only.

- **Customer objects path** is the path to the currently used custom objects library which you can manage yourself. You can change the path using an external tool. Refer to How to Change the Location of the Custom Objects Library (MotionDesk Custom Object Library Management 🕮).

Related topics

Basics

3-D Object Libraries (MotionDesk Scene Creation 🕮)
Working with Observers in Zone Navigation Mode..... 96

References

MotionDesk Options (MotionDesk Basics 🕮)

Automation

Classes for Scene Animation

Introduction

The following topics provides a description of the classes that you can use for automating the animation in MotionDesk.

Where to go from here

Information in this section

Information in other sections

Introduction to the MotionDesk Automation Interface (MotionDesk Automation 🕮)
Introduces the MotionDesk automation interface and the required user experience.

Features of MotionDesk Automation Interface (MotionDesk Automation 🕮)
Describes the features of the MotionDesk automation interface.

Overview of the Object Model (MotionDesk Automation 🕮)
Shows you the object dependencies, object attributes and methods in the MotionDesk object model at a glance.

Example of Automating MotionDesk with a Python Script (MotionDesk Automation 🕮)
Code examples demonstrate how you can automate MotionDesk with a Python script.

AtmosphericsManager

Purpose

To get or set the atmospherics settings.

Class Description (AtmosphericsManager)

Syntax

```
VisualizationManager = Experiment.VisualizationManagement
Atmospherics = VisualizationManager.Atmospherics
```

Purpose

To get or set the atmospherics settings.

Attributes

The class contains the following attributes:

Attributes	Type	Purpose
ActiveAtmosphericsMode	AtmosphericsModes[1)]	To get/set the atmospherics mode.
CustomAtmospherics	CustomAtmospherics[2)]	To get the settings of the custom atmospherics.

1) Refer to AtmosphericsModes on page 276.
2) Refer to CustomAtmospherics on page 221.

Methods

–

Related topics

References

Class Description (Experiment) (MotionDesk Project and Experiment Management 🕮)

CustomAtmospherics

Purpose

To specify the settings of the custom atmospherics.

Where to go from here

Information in this section

Class Description (CustomAtmospherics)

Syntax

```
CustomAtmospherics = AtmosphericsManager.CustomAtmosphericSettings
```

Purpose

To specify the settings of the custom atmospherics.

Attributes

The class contains the following attributes:

Attributes	Type	Purpose
HeadlightIntensity	Double	To get/set the intensity of the head light.
Sunlight	Sunlight[1)]	To get the sunlight settings.
Fog	Fog[2)]	To get the fog settings.
Rain	Rain[3)]	To get the rain settings.
Snow	Snow[4)]	To get the snow settings.
Shadow	Shadow[5)]	To get the shadow settings.

1) Refer to Sunlight on page 273.
2) Refer to Fog on page 223.
3) Refer to Rain on page 233.
4) Refer to Snow on page 273.
5) Refer to Shadow on page 234.

Methods

The class contains the following methods:

Method	Purpose
ApplyPreset	To set a preset mode for custom atmospherics. Refer to ApplyPreset on page 222.

Related topics

References

Class Description (AtmosphericsManager)..... 220

ApplyPreset

Class

CustomAtmospherics

Syntax

```
CustomAtmospherics.ApplyPreset(AtmosphericPresetModes PresetMode)
```

Purpose

To set a preset mode for custom atmospherics.

Parameters

The method uses the following parameters:

Parameter	Type	Description
PresetMode	AtmosphericPresetModes[1)]	Preset mode

[1)] Refer to AtmosphericPresetModes on page 276

Return value

–

Related topics

References

Class Description (CustomAtmospherics)..... 221

DataStreamNames

Purpose

To get information on the data stream.

Class Description (DataStreamNames)

Syntax

```
SimulationManager = Application.SimulationManagement
DataStreams = SimulationManager.DataStreamNames
```

Purpose

To get information on the data stream.

Attributes

The class contains the following attributes:

Attributes	Type	Purpose
BodyNames	String[]	To get the names of the movable objects.
SignalNames	String[]	To get the names of the signals.
SignalUnits	String[]	To get the units of the signals.

Methods

–

Related topics

References

Class Description (SimulationManager)..... 269

Fog

Purpose

To get or set the settings of a fog atmospherics.

Class Description (Fog)

Syntax

```
Fog = CustomAtmosphericsSettings.Fog
```

Purpose

To get or set the settings of a fog atmospherics.

Attributes

The class contains the following attributes:

Attributes	Type	Purpose
Enabled	Boolean	To get/set the switch for enabling the fog.
Color	Integer[]	To get/set the color as a tuple of 3 integer value (RGB).
Range	Double	To get/set the range.
Density	DensityModes[1].	To get/set the density mode.

[1] Refer to DensityModes on page 276

Methods

–

Related topics

References

Atmospherics Properties..... 189
Class Description (CustomAtmospherics)..... 221

Observer

Purpose

To specify an observer.

Where to go from here

Information in this section

Class Description (Observer)..... 225
To describe the class and its attributes.

GetPositionAndOrientation..... 226
To get the position and orientation of the observer.

SetOrientation..... 227
To set the orientation of the observer.

SetPosition..... 227
To set the position of the observer.

SetPositionAndOrientation..... 228
To set the position and orientation of the observer.

Class Description (Observer)

Syntax

```
Observer = ObserverManager.Item(objectIdentifier)
```

or

```
Observer = ObserverManager.Add(observerName)
```

Purpose

To specify an observer.

Attributes

The class contains the following attributes:

Attributes	Type	Purpose
AttachedTo	string	To get/set the movable object when the observer is to follow a movable object.
Behaviour	FollowBehaviour[1)]	To get/set the behavior of the observer.
GetOrientation	double[]	To get all angles of the orientation.
GetPosition	double[]	To get all coordinates of the position.
Name	string	To get/set the name of the observer.
XOrientation	double	To get/set angle of rotation (x-axis) of the observer's orientation.
XPosition	double	To get/set the x coordinate of the observer's position.
YOrientation	double	To get/set angle of elevation (y-axis) of the observer's orientation.
YPosition	double	To get/set the y coordinate of the observer's position.
ZOrientation	double	To get/set the angle of azimuth (z-axis) of the observer's orientation.
ZPosition	double	To get/set the z coordinate of the observer's position.

[1)] Refer to FollowBehaviour on page 276.

Methods

The class contains the following methods:

Method	Purpose
GetPositionAndOrientation	To get the position and orientation of the observer. Refer to GetPositionAndOrientation on page 226.
SetOrientation	To set the orientation of the observer. Refer to SetOrientation on page 227.
SetPosition	To set the position of the observer. Refer to SetPosition on page 227.
SetPositionAndOrientation	To set the position and orientation of the observer. Refer to SetPositionAndOrientation on page 228.

Related topics

References

Class Description (ObserverManager)..... 229

GetPositionAndOrientation

Class

Observer

Syntax

```
RetVal = Observer.GetPositionAndOrientation(Position, Orientation)
```

Purpose

To get the position and orientation of the observer.

Parameters

The method uses the following parameters:

Parameter	Type	Description
Position	double[]	Position of the observer (x, y, z coordinates)
Orientation	double[]	Orientation of the observer (angles around x-, y-, z-axis)

Return value

The method returns the following parameter:

Type	Description
Boolean	True if successful

Related topics

References

Class Description (Observer)..... 225

SetOrientation

Class

Observer

Syntax

```
Observer.SetOrientation(Orientation)
```

Purpose

To set the orientation of the observer.

Parameters

The method uses the following parameters:

Parameter	Type	Description
Orientation	double[]	Orientation of the observer (angles around x-, y-, z-axis)

Return value

–

Related topics

References

Class Description (Observer)..... 225

SetPosition

Class

Observer

Syntax

```
Observer.SetPosition(Position)
```

Purpose

To set the position of the observer.

Parameters

The method uses the following parameters:

Parameter	Type	Description
Position	double[]	Position of the observer (x, y, z coordinates)

Return value

–

Related topics

References

Class Description (Observer)..... 225

SetPositionAndOrientation

Class

Observer

Syntax

```
Observer.SetPositionAndOrientation(Position, Orientation)
```

Purpose

To set the position and orientation of the observer.

Parameters

The method uses the following parameters:

Parameter	Type	Description
Position	double[]	Position of the observer (x, y, z coordinates)
Orientation	double[]	Orientation of the observer (angles around x-, y-, z-axis)

Return value

–

Related topics

References

Class Description (Observer)..... 225

ObserverManager

Purpose

To handle the observer objects of a MotionDesk experiment.

Where to go from here

Information in this section

Class Description (ObserverManager)

Syntax

```
ObserverManager = VisualizationManager.Observers
```

Purpose

To handle the observer.

Attributes

The class contains the following attributes:

Attributes	Type	Purpose
Count	Integer	To get the number of observers.

Methods

The class contains the following methods:

Method	Purpose
Add	To add a new observer. Refer to Add on page 230.
CreateDefaultObservers	To create default observers. Refer to CreateDefaultObservers on page 231.
Item	To access an observer. Refer to Item on page 231.

Method	Purpose
Remove	To remove an observer. Refer to Remove on page 232.

Related topics

References

Class Description (VisualizationManager)..... 274

Add

Class

ObserverManager

Syntax

```
Observer = ObserverManager.Add(observerName)
```

Purpose

To add a new observer.

Parameters

The method uses the following parameters:

Parameter	Type	Description
observerName	string	The name of the new observer.

Return value

The method returns the following parameter:

Type	Description
Observer[1)]	The new observer object.

[1)] Refer to Observer on page 224.

Related topics

References

Class Description (ObserverManager)..... 229

CreateDefaultObservers

Class

ObserverManager

Syntax

```
RetVal = ObserverManager.CreateDefaultObservers(movableName)
```

Purpose

To create default observers.

Parameters

The method uses the following parameters:

Parameter	Type	Description
movableName	string	Name of the movable object to which the default observer is attached.

Return value

The method returns the following parameter:

Type	Description
Boolean	True if successful.

Related topics

References

Class Description (ObserverManager)..... 229

Item

Class

ObserverManager

Syntax

```
Observer = ObserverManager.Item(objectIdentifier)
```

Purpose

To access an observer.

Parameters

The method uses the following parameters:

Parameter	Type	Description
objectIdentifier	object	Identifier of the observer. The identifier can be specified as string (observer name) or integer (index).

Return value

The method returns the following parameter:

Type	Description
Observer[1)]	The specific observer object.

1) Refer to Observer on page 224.

Related topics

References

Class Description (ObserverManager)..... 229

Remove

Class

ObserverManager

Syntax

```
RetVal = ObserverManager.Remove(observerName)
```

Purpose

To remove an observer.

Parameters

The method uses the following parameters:

Parameter	Type	Description
observerName	string	The name of the new observer.

Return value

The method returns the following parameter:

Type	Description
Boolean	True if successful.

Related topics

References

Class Description (ObserverManager)..... 229

Rain

Purpose

To get or set the settings of rain scenarios.

Class Description (Rain)

Syntax

```
Rain = CustomAtmosphericsSettings.Rain
```

Purpose

To get or set the settings of rain scenarios.

Attributes

The class contains the following attributes:

Attributes	Type	Purpose
Enabled	Boolean	To get/set the switch for enabling the rain.
Color	Integer[]	To get/set the color as a tuple of 3 integer value (RGB).
Speed	Double	To get/set the speed.
Intensity	Double	To get/set the intensity.
ParticelSize	Double	To get/set the particle size.
FallingAngle	Double	To get/set the falling angle.
HeadingAngle	Double	To get/set the heading angle.

Methods

–

Related topics

References

Atmospherics Properties..... 189
Class Description (CustomAtmospherics)..... 221

Shadow

Purpose

To get or set the settings of shadow.

Class Description (Shadow)

Syntax

```
Shadow = CustomAtmosphericsSettings.Shadow
```

Purpose

To get or set the settings of shadow.

Attributes

The class contains the following attributes:

Attributes	Type	Purpose
Enabled	Boolean	To get/set the switch for enabling the shadow.
Transparency	Double	To get/set the transparency.

Methods

–

Related topics

References

Atmospherics Properties 189
Class Description (CustomAtmospherics) 221

SimulationBufferFrameData

Purpose

To access the frame data of the simulation buffer.

Where to go from here

Information in this section

Class Description (SimulationBufferFrameData) 235
To describe the class and its attributes.

Class Description (SimulationBufferFrameData)

Syntax

Is not created directly

```
SimulationManager = Application.SimulationManagement
SimulationBuffer = SimulationManager.SimulationBuffer
FrameData = SimulationBuffer.FrameData
```

Purpose

To access the frame data of the simulation buffer.

Attributes

The class contains the following attributes:

Attributes	Type	Purpose
Bodies	Integer	To get the number of bodies.
Signals	Integer	To get the number of signals.
Time	Double	To get the time stamp.

Methods

The class contains the following methods:

Method	Purpose
MotionData	To access the motion data of a body. Refer to MotionData on page 236.
SignalData	To get a signal data. Refer to SignalData on page 236.

Related topics

References

Class Description (SimulationBufferManager)..... 238

MotionData

Class

SimulationBufferFrameData

Syntax

```
RetVal = MySimulationBufferFrameData.MotionData(bodyIndex)
```

Purpose

To access the motion data of a body.

Parameters

The method uses the following parameters:

Parameter	Type	Description
bodyIndex	Integer	Index of a body.

Return value

The method returns the following parameter:

Type	Description
SimulationMotionData[1)]	Access to the motion data of the selected body.

[1)] Refer to SimulationMotionData on page 272.

Related topics

References

Class Description (SimulationBufferFrameData)..... 235

SignalData

Class

SimulationBufferFrameData

Syntax

```
RetVal = MySimulationBufferFrameData.SignalData(signalIndex)
```

Purpose

To access the data of a signal.

Parameters

The method uses the following parameters:

Parameter	Type	Description
signalIndex	Integer	Index of a signal

Return value

The method returns the following parameter:

Type	Description
Double	Values of the signal.

Related topics

References

Class Description (SimulationBufferFrameData)..... 235

SimulationBufferManager

Purpose

To get information on the simulation buffer.

Where to go from here

Information in this section

Class Description (SimulationBufferManager)..... 238
To describe the class and its attributes.

IsPlaying..... 238
To check whether the animation is running.

Play..... 239
To start the animation.

StopPlaying..... 240
To stop the animation.

StoreBufferedFrames..... 240
To store the buffered frames to an MDF file.

Class Description (SimulationBufferManager)

Syntax

```
SimulationManager = Application.SimulationManagement
SimulationBuffer = SimulationManager.SimulationBuffer
```

Purpose

To get information on the simulation buffer.

Attributes

The class contains the following attributes:

Attributes	Type	Purpose
Frame	Integer	To get/set the number of the frame.
FrameData	SimulationBufferFrameData[1)]	To get the frame data.
IsEmpty	Boolean	To get information whether the buffer is empty.

[1)] Refer to SimulationBufferFrameData on page 234.

Methods

The class contains the following methods:

Method	Purpose
IsPlaying	To check whether the animation is running. Refer to IsPlaying on page 238.
Play	To start the animation. Refer to Play on page 239.
StopPlaying	To stop the animation. Refer to StopPlaying on page 240.
StoreBufferedFrames	To store the buffered frames to an MDF file. Refer to StoreBufferedFrames on page 240.

Related topics

References

Class Description (Application) (MotionDesk Basics 🕮)
Class Description (SimulationManager)..... 269

IsPlaying

Class

SimulationBufferManager

Syntax

```
MySimulationBufferManager.IsPlaying()
```

Purpose To check whether the animation is running.

Parameters –

Return value The method returns the following parameter:

Type	Description
Boolean	True if the animation is running.

Related topics

References

Class Description (SimulationBufferManager)..... 238

Play

Class SimulationBufferManager

Syntax

```
MySimulationBufferManager.Play()
```

Purpose To start the animation.

Parameters –

Return value –

Related topics

References

Class Description (SimulationBufferManager)..... 238

StopPlaying

Class	SimulationBufferManager
Syntax	`MySimulationBufferManager.StopPlaying()`
Purpose	To stop the animation.
Parameters	–
Return value	–
Related topics	References Class Description (SimulationBufferManager)..... 238

StoreBufferedFrames

Class	SimulationBufferManager
Syntax	`MySimulationBufferManager.StoreBufferedFrames(mdfFilePath)`
Purpose	To store the buffered frames to an MDF file.
Parameters	The method uses the following parameters:

Parameter	Type	Description
mdfFilePath	String	Name and file path of the MDF file.

Return value	–

Related topics

References

Class Description (SimulationBufferManager)..... 238

SimulationDataPointADAS

Purpose

To specify the settings of an ADAS simulator to be used as the data source.

Class Description (SimulationDataPointADAS)

Syntax

```
SDPADAS = SimulationManager.SimulationDataPointADAS
```

Purpose

To specify the settings of an ADAS simulator to be used as the data source.

Description

The `SimulationDataPointADAS` class specifies a network adapter as the data source. To actually use it as data source, you must activate the `ADAS` as simulation data source in the `ActiveSimulationDataPoint` attribute of the `SimulationManagement` object. Refer to Class Description (SimulationManager) on page 269.

Attributes

The class contains the following attributes:

Attributes	Type	Purpose
IPv4Address	String	To get/set the IP address of the ADAS Simulator.
Port	Integer	To get/set the TCP port number for the port that is used to receive motion data from the simulator.

Methods

–

Related topics

References

Class Description (SimulationManager)..... 269

SimulationDataPointFile

Purpose

To specify the settings of an MDF file to be used as the data source.

Class Description (SimulationDataPointFile)

Syntax

```
SimulationManager = Application.SimulationManagement
SDPFile = SimulationManager.SimulationDataPointFile
```

Purpose

To specify the settings of an MDF file to be used as the data source.

Description

The `SimulationDataPointFile` class specifies an MDF file as data source. To actually use an MDF file as the data source, you must activate the `File` as simulation data source in the `ActiveSimulationDataPoint` attribute of the `SimulationManagement` object.

Attributes

The class contains the following attributes:

Attributes	Type	Purpose
MotionDataFilePath	String	To get/set the name and path of the MDF file.

Methods

–

Related topics

References

Class Description (SimulationManager)..... 269

SimulationDataPointHardware

Purpose

To specify the settings of a dSPACE hardware platform connected via bus connection to be used as the data source.

Where to go from here

Information in this section

Class Description (SimulationDataPointHardware)

Syntax

```
SDPHardware = SimulationManager.SimulationDataPointHardware
```

Purpose

To specify the settings of a dSPACE hardware platform connected via bus connection to be used as the data source.

Description

The `SimulationDataPointHardware` class specifies a dSPACE hardware platform connected via bus connection as data source. To actually use the platform as data source, you must activate the `Hardware` as simulation data source in the `ActiveSimulationDataPoint` attribute of the `SimulationManagement` object.

Attributes

The class contains the following attributes:

Attributes	Type	Purpose
DaqRefreshInterval	Integer	To get/set the DAQ refresh interval.
Stream	String	To get the real-time board that is used as data source.
StreamBufferSize	Integer	To get/set the number of frames that is buffered during animation.
Type	PHSHardware[1)]	To get the board type.

[1)] Refer to PHSHardware on page 277.

Methods

The class contains the following methods:

Method	Purpose
Connect	To connect MotionDesk to the real-time application. Refer to Connect on page 244.

Related topics

References

Class Description (SimulationManager)..... 269

Connect

Class

SimulationDataPointHardware

Syntax

```
SimulationDataPointHardware.Connect(hardwareType, streamName)
```

Purpose

To connect MotionDesk to the real-time application.

Parameters

The method uses the following parameters:

Parameter	Type	Description
hardwareType	PHSHardware[1)]	Specifies the board type
streamName	String	Specifies the name of the real-time board

1) Refer to PHSHardware on page 277.

Return value

The method returns the following parameter:

Type	Description
Boolean	True if successful connected.

Related topics

References

Class Description (SimulationDataPointHardware)..... 243

SimulationDataPointHardwareNet

Purpose

To specify the settings of a dSPACE hardware platform connected via network connection to be used as data source.

Where to go from here

Information in this section

Class Description (SimulationDataPointHardwareNet)

Syntax

```
SDPHardwareNet = SimulationManager.SimulationDataPointHardwareNet
```

Purpose

To specify the settings of a dSPACE hardware platform connected via network connection to be used as the data source.

Description

The `SimulationDataPointHardwareNet` class specifies a dSPACE hardware platform connected via network connection as the data source. To actually use the platform as data source, you must activate the `HardwareNet` as simulation data source in the `ActiveSimulationDataPoint` attribute of the `SimulationManagement` object.

Attributes

The class contains the following attributes:

Attributes	Type	Purpose
Stream	String	To get the real-time board that is used as data source.
StreamBufferSize	Integer	To get/set the number of frames that is buffered during animation.
Type	PHSHardware[1)]	To get the board type.

1) Refer to PHSHardware on page 277.

Methods

The class contains the following methods:

Method	Purpose
Connect	To connect MotionDesk to the real-time application. Refer to Connect on page 246.

Related topics

References

Class Description (SimulationManager)..... 269

Connect

Class

SimulationDataPointHardwareNet

Syntax

```
SimulationDataPointHardwareNet.Connect(hardwareType, streamName)
```

Purpose

To connect MotionDesk to the real-time application.

Parameters

The method uses the following parameters:

Parameter	Type	Description
hardwareType	PHSHardware[1)]	Specifies the board type.
streamName	String	Specifies the name of the real-time board.

1) Refer to PHSHardware on page 277.

Return value

The method returns the following parameter:

Type	Description
Boolean	True if successful connected.

Related topics

References

Class Description (SimulationDataPointHardwareNet)..... 245

SimulationDataPointMlbx

Purpose

To specify the settings of MicroLabBox to be used as the data source.

Where to go from here

Information in this section

Class Description (SimulationDataPointMlbx)

Syntax

```
SDPMlbx = SimulationManager.SimulationDataPointMlbx
```

Purpose

To specify the settings of MicroLabBox to be used as the data source.

Description

The `SimulationDataPointMlbx` class specifies MicroLabBox as the data source. To actually use the platform as data source, you must activate the `Mlbx` as simulation data source in the `ActiveSimulationDataPoint` attribute of the `SimulationManagement` object.

Attributes

The class contains the following attributes:

Attributes	Type	Purpose
BoardSystemName	String	To get the name of MicroLabBox.
IPv4Address	String	To get the IP address of MicroLabBox.
ApplicationName	String	To get the name of the application that sends the motion data.

Methods

The class contains the following methods:

Method	Purpose
AssignBoardByIP	To assign MicroLabBox by using its IP address. Refer to AssignBoardByIP on page 248.
AssignBoardByName	To assign MicroLabBox by using its name. Refer to AssignBoardByName on page 249.
UnassignBoard	To unassign MicroLabBox currently used as data source. Refer to UnassignBoard on page 249.
ConnectToApplication	To connect to the application which sends the motion data. Refer to ConnectToApplication on page 250.

Related topics

References

Class Description (SimulationManager)..... 269

AssignBoardByIP

Class

SimulationDataPointMlbx

Syntax

```
SimulationDataPointMlbx.AssignBoardByIP(ipAddress)
```

Purpose

To assign MicroLabBox by using its IP address.

Parameters

The method uses the following parameters:

Parameter	Type	Description
ipAddress	String	IP address of MicroLabBox

Return value

The method returns the following parameter:

Type	Description
Boolean	True if successful assigned.

Related topics

References

Class Description (SimulationDataPointMlbx)..... 247

AssignBoardByName

Class

SimulationDataPointMlbx

Syntax

```
SimulationDataPointMlbx.AssignBoardByName(systemName)
```

Purpose

To assign MicroLabBox by using its name.

Parameters

The method uses the following parameters:

Parameter	Type	Description
systemName	String	Name of MicroLabBox

Return value

The method returns the following parameter:

Type	Description
Boolean	True if successful assigned.

Related topics

References

Class Description (SimulationDataPointMlbx)..... 247

UnassignBoard

Class

SimulationDataPointMlbx

Syntax

```
SimulationDataPointMlbx.UnassignBoard()
```

Purpose

To unassign MicroLabBox currently used as data source.

Parameters

–

Return value

The method returns the following parameter:

Type	Description
Boolean	True if successful unassigned.

Related topics

References

Class Description (SimulationDataPointMlbx)..... 247

ConnectToApplication

Class

SimulationDataPointMlbx

Syntax

```
RetVal = ConnectToApplication(String ApplicationName)
```

Purpose

To connect to the application which sends the motion data.

Parameters

The method uses the following parameters:

Parameter	Type	Description
ApplicationName	String	Name of the application which sends the motion data.

Return value

The method returns the following parameter:

Type	Description
Boolean	True if successful.

Related topics

References

Class Description (SimulationDataPointMlbx)..... 247

SimulationDataPointMABX

Purpose

To specify the settings of MicroAutoBox III to be used as the data source.

Where to go from here

Information in this section

Class Description (SimulationDataPointMABX)

Syntax

```
SDPMABX = SimulationManager.SimulationDataPointMABX
```

Purpose

To specify the settings of MicroAutoBox III to be used as the data source.

Description

The `SimulationDataPointMABX` class specifies MicroAutoBox III as the data source. To actually use the platform as data source, you must activate the `MABX` as simulation data source in the `ActiveSimulationDataPoint` attribute of the `SimulationManagement` object.

Attributes

The class contains the following attributes:

Attributes	Type	Purpose
BoardSystemName	String	To get the name of MicroAutoBox III.
IPv4Address	String	To get the IP address of MicroAutoBox III.
ApplicationName	String	To get the name of the application that sends the motion data.

Methods

The class contains the following methods:

Method	Purpose
AssignBoardByIP	To assign MicroAutoBox III by using its IP address. Refer to AssignBoardByIP on page 252.
AssignBoardByName	To assign MicroAutoBox III by using its name. Refer to AssignBoardByName on page 253.
UnassignBoard	To unassign MicroAutoBox III currently used as data source. Refer to UnassignBoard on page 254.
ConnectToApplication	To connect to the application which sends the motion data. Refer to ConnectToApplication on page 254.

Related topics

References

Class Description (SimulationManager)..... 269

AssignBoardByIP

Class

SimulationDataPointMABX

Syntax

```
SimulationDataPointMABX.AssignBoardByIP(ipAddress)
```

Purpose

To assign MicroAutoBox III by using its IP address.

Parameters

The method uses the following parameters:

Parameter	Type	Description
ipAddress	String	IP address of MicroAutoBox III

Return value

The method returns the following parameter:

Type	Description
Boolean	True if successful assigned.

Related topics

References

Class Description (SimulationDataPointMABX)..... 251

AssignBoardByName

Class

SimulationDataPointMABX

Syntax

```
SimulationDataPointMABX.AssignBoardByName(systemName)
```

Purpose

To assign MicroAutoBox III by using its name.

Parameters

The method uses the following parameters:

Parameter	Type	Description
systemName	String	Name of MicroAutoBox III

Return value

The method returns the following parameter:

Type	Description
Boolean	True if successful assigned.

Related topics

References

Class Description (SimulationDataPointMABX)..... 251

UnassignBoard

Class

SimulationDataPointMABX

Syntax

```
SimulationDataPointMABX.UnassignBoard()
```

Purpose

To unassign MicroAutoBox III currently used as data source.

Parameters

–

Return value

The method returns the following parameter:

Type	Description
Boolean	True if successful unassigned.

Related topics

References

Class Description (SimulationDataPointMABX)..... 251

ConnectToApplication

Class

SimulationDataPointMABX

Syntax

```
RetVal = ConnectToApplication(String ApplicationName)
```

Purpose

To connect to the application which sends the motion data.

Parameters

The method uses the following parameters:

Parameter	Type	Description
ApplicationName	String	Name of the application which sends the motion data.

Return value

The method returns the following parameter:

Type	Description
Boolean	True if successful.

Related topics

References

Class Description (SimulationDataPointMABX)..... 251

SimulationDataPointNetwork

Purpose

To specify the settings of a network adapter to be used as the data source.

Class Description (SimulationDataPointNetwork)

Syntax

```
SDPNetwork = SimulationManager.SimulationDataPointNetwork
```

Purpose

To specify the settings of a network adapter to be used as the data source.

Description

The `SimulationDataPointNetwork` class specifies a network adapter as the data source. To actually use it as data source, you must activate the `Network` as simulation data source in the `ActiveSimulationDataPoint` attribute of the `SimulationManagement` object. Refer to Class Description (SimulationManager) on page 269.

Attributes

The class contains the following attributes:

Attributes	Type	Purpose
IPv4Address	String	To get/set the IP address of the MotionDesk host PC.
Port1	Integer	To get/set the UDP port number for the port that is used to receive motion data from the simulator.
Port2	Integer	To get/set the UDP port number for the port that is used to send configuration requests from the MotionDesk PCs to the simulator.

Attributes	Type	Purpose
Port3	Integer	To get/set the UDP port number for the port that is used to receive configuration responses from the simulator.
StreamBufferSize	Integer	To get/set the number of frames that are buffered by streaming.

Methods

–

Related topics

References

Class Description (SimulationManager)..... 269

SimulationDataPointScalexio

Purpose

To specify the settings of a SCALEXIO platform to be used as the data source.

Where to go from here

Information in this section

Class Description (SimulationDataPointScalexio)..... 256
To describe the class and its attributes.

AssignBoardByIP..... 258
To assign a SCALEXIO system by using its IP address.

AssignBoardByName..... 258
To assign a SCALEXIO system by using its name.

UnassignBoard..... 259
To unassign the SCALEXIO system currently used as data source.

ConnectToApplication..... 260
To connect to the application which sends the motion data.

Class Description (SimulationDataPointScalexio)

Syntax

```
SDPScalexio = SimulationManager.SimulationDataPointScalexio
```

Purpose

To specify the settings of a SCALEXIO platform to be used as the data source.

Description

The `SimulationDataPointScalexio` class specifies a SCALEXIO platform as the data source. To actually use the platform as data source, you must activate the `SCALEXIO` as simulation data source in the `ActiveSimulationDataPoint` attribute of the `SimulationManagement` object.

Attributes

The class contains the following attributes:

Attributes	Type	Purpose
BoardSystemName	String	To get the name of the SCALEXIO platform.
IPv4Address	String	To get the IP address of the SCALEXIO platform.
StreamBufferSize	Integer	To get/set the number of frames that is buffered during animation.
ApplicationName	String	To get the name of the application that sends the motion data.

Methods

The class contains the following methods:

Method	Purpose
AssignBoardByIP	To assign a SCALEXIO platform by using its IP address. Refer to AssignBoardByIP on page 258.
AssignBoardByName	To assign a SCALEXIO platform by using its name. Refer to AssignBoardByName on page 258.
UnassignBoard	To unassign the SCALEXIO platform currently used as data source. Refer to UnassignBoard on page 259.
ConnectToApplication	To connect to the application which sends the motion data. Refer to ConnectToApplication on page 260.

Related topics

References

Class Description (SimulationManager)..... 269

AssignBoardByIP

Class

SimulationDataPointScalexio

Syntax

```
SimulationDataPointScalexio.AssignBoardByIP(ipAddress)
```

Purpose

To assign a SCALEXIO system by using its IP address.

Parameters

The method uses the following parameters:

Parameter	Type	Description
ipAddress	String	IP address of the SCALEXIO system

Return value

The method returns the following parameter:

Type	Description
Boolean	True if successful assigned.

Related topics

References

Class Description (SimulationDataPointScalexio)..... 256

AssignBoardByName

Class

SimulationDataPointScalexio

Syntax

```
SimulationDataPointScalexio.AssignBoardByName(systemName)
```

Purpose

To assign a SCALEXIO system by using its name.

Parameters

The method uses the following parameters:

Parameter	Type	Description
systemName	String	Name of the SCALEXIO system

Return value

The method returns the following parameter:

Type	Description
Boolean	True if successful assigned.

Related topics

References

Class Description (SimulationDataPointScalexio)..... 256

UnassignBoard

Class

SimulationDataPointScalexio

Syntax

```
SimulationDataPointScalexio.UnassignBoard()
```

Purpose

To unassign the SCALEXIO system currently used as data source.

Parameters

–

Return value

The method returns the following parameter:

Type	Description
Boolean	True if successful unassigned.

Related topics

References

Class Description (SimulationDataPointScalexio)..... 256

ConnectToApplication

Class

SimulationDataPointScalexio

Syntax

```
RetVal = ConnectToApplication(String ApplicationName)
```

Purpose

To connect to the application which sends the motion data.

Parameters

The method uses the following parameters:

Parameter	Type	Description
ApplicationName	String	Name of the application which sends the motion data.

Return value

The method returns the following parameter:

Type	Description
Boolean	True if successful.

Related topics

References

Class Description (SimulationDataPointScalexio)..... 256

SimulationDataPointVEOS

Purpose

To specify the settings of a VEOS to be used as the data source.

Where to go from here

Information in this section

Class Description (SimulationDataPointVEOS)..... 261
To describe the class and its attributes.

AssignBoardByIP..... 262
To assign a VEOS by using its IP address.

AssignBoardByName..... 262
To assign a VEOS by using its name.

Class Description (SimulationDataPointVEOS)

Syntax

```
SDPVeos = SimulationManager.SimulationDataPointVeos
```

Purpose

To specify the settings of a VEOS to be used as the data source.

Description

The `SimulationDataPointVeos` class specifies a VEOS as the data source. To actually use the platform as data source, you must activate the `VEOS` as simulation data source in the `ActiveSimulationDataPoint` attribute of the `SimulationManagement` object.

Attributes

The class contains the following attributes:

Attributes	Type	Purpose
BoardSystemName	String	To get the name of the VEOS.
IPv4Address	String	To get the IP address of the VEOS.
StreamBufferSize	Integer	To get/set the number of frames that is buffered during animation.
ApplicationName	String	To get the name of the application that sends the motion data.

Methods

The class contains the following methods:

Method	Purpose
AssignBoardByIP	To assign a VEOS by using its IP address. Refer to AssignBoardByIP on page 262.
AssignBoardByName	To assign a VEOS by using its name. Refer to AssignBoardByName on page 262.
UnassignBoard	To unassign the VEOS currently used as data source. Refer to UnassignBoard on page 263.
ConnectToApplication	To connect to the application which sends the motion data. Refer to ConnectToApplication on page 264.

Related topics

References

Class Description (SimulationManager)..... 269

AssignBoardByIP

Class

SimulationDataPointVEOS

Syntax

```
SimulationDataPointVEOS.AssignBoardByIP(ipAddress)
```

Purpose

To assign a VEOS by using its IP address.

Parameters

The method uses the following parameters:

Parameter	Type	Description
ipAddress	String	IP address of the VEOS

Return value

The method returns the following parameter:

Type	Description
Boolean	True if successful assigned.

Related topics

References

Class Description (SimulationDataPointVEOS)..... 261

AssignBoardByName

Class

SimulationDataPointVEOS

Syntax

```
SimulationDataPointVEOS.AssignBoardByName(systemName)
```

Purpose

To assign a VEOS by using its name.

Parameters

The method uses the following parameters:

Parameter	Type	Description
systemName	String	Name of the VEOS

Return value

The method returns the following parameter:

Type	Description
Boolean	True if successful assigned.

Related topics

References

Class Description (SimulationDataPointVEOS)..... 261

UnassignBoard

Class

SimulationDataPointVEOS

Syntax

```
SimulationDataPointVEOS.UnassignBoard()
```

Purpose

To unassign the VEOS currently used as data source.

Parameters

–

Return value

The method returns the following parameter:

Type	Description
Boolean	True if successful unassigned.

Related topics

References

Class Description (SimulationDataPointVEOS)..... 261

ConnectToApplication

Class

SimulationDataPointVEOS

Syntax

```
RetVal = ConnectToApplication(String ApplicationName)
```

Purpose

To connect to the application which sends the motion data.

Parameters

The method uses the following parameters:

Parameter	Type	Description
ApplicationName	String	Name of the application which sends the motion data.

Return value

The method returns the following parameter:

Type	Description
Boolean	True if successful.

Related topics

References

Class Description (SimulationDataPointVEOS)..... 261

SimulationDataPointDS1007

Purpose

To specify the settings of a DS1007 platform to be used as the data source.

Where to go from here

Information in this section

Class Description (SimulationDataPointDS1007)

Syntax

```
SDPDS1007 = SimulationManager.SimulationDataPointDS1007
```

Purpose

To specify the settings of a DS1007 platform to be used as the data source.

Description

The `SimulationDataPointDS1007` class specifies a DS1007 platform as the data source. To actually use the platform as data source, you must activate the `DS1007` as simulation data source in the `ActiveSimulationDataPoint` attribute of the `SimulationManagement` object.

Attributes

The class contains the following attributes:

Attributes	Type	Purpose
BoardSystemName	String	To get the name of the DS1007.
IPv4Address	String	To get the IP address of the DS1007 board.
StreamBufferSize	Integer	To get/set the number of frames that is buffered during animation.
ApplicationName	String	To get the name of the application that sends the motion data.

Methods

The class contains the following methods:

Method	Purpose
AssignBoardByIP	To assign a DS1007 by using its IP address. Refer to AssignBoardByIP on page 266.
AssignBoardByName	To assign a DS1007 by using its name. Refer to AssignBoardByName on page 267.
UnassignBoard	To unassign the DS1007 currently used as data source. Refer to UnassignBoard on page 267.
ConnectToApplication	To connect to the application which sends the motion data. Refer to ConnectToApplication on page 268.

Related topics

References

Class Description (SimulationManager)..... 269

AssignBoardByIP

Class

SimulationDataPointDS1007

Syntax

```
SimulationDataPointDS1007.AssignBoardByIP(ipAddress)
```

Purpose

To assign a DS1007 by using its IP address.

Parameters

The method uses the following parameters:

Parameter	Type	Description
ipAddress	String	IP address of the DS1007

Return value

The method returns the following parameter:

Type	Description
Boolean	True if successful assigned.

Related topics

References

Class Description (SimulationDataPointDS1007)..... 265

AssignBoardByName

Class

SimulationDataPointDS1007

Syntax

```
SimulationDataPointDS1007.AssignBoardByName(systemName)
```

Purpose

To assign a DS1007 by using its name.

Parameters

The method uses the following parameters:

Parameter	Type	Description
systemName	String	Name of the DS1007

Return value

The method returns the following parameter:

Type	Description
Boolean	True if successful assigned.

Related topics

References

Class Description (SimulationDataPointDS1007)..... 265

UnassignBoard

Class

SimulationDataPointDS1007

Syntax

```
SimulationDataPointDS1007.UnassignBoard()
```

Purpose

To unassign the DS1007 currently used as data source.

Parameters

–

Return value

The method returns the following parameter:

Type	Description
Boolean	True if successful unassigned.

Related topics

References

Class Description (SimulationDataPointDS1007)..... 265

ConnectToApplication

Class

SimulationDataPointDS1007

Syntax

```
RetVal = ConnectToApplication(String ApplicationName)
```

Purpose

To connect to the application which sends the motion data.

Parameters

The method uses the following parameters:

Parameter	Type	Description
ApplicationName	String	Name of the application which sends the motion data.

Return value

The method returns the following parameter:

Type	Description
Boolean	True if successful.

Related topics

References

Class Description (SimulationDataPointDS1007)..... 265

SimulationManager

Purpose

To access simulation management.

Where to go from here

Information in this section

Class Description (SimulationManager)..... 269
To describe the class and its attributes.

StartAnimation..... 271
To start the animation.

StopAnimation..... 271
To stop the animation.

Class Description (SimulationManager)

Syntax

```
SimulationManager = Application.SimulationManagement
```

Purpose

To access simulation management.

Attributes

The class contains the following attributes:

Attributes	Type	Purpose
ActiveSimulationDataPoint	SimulationDataSources[1)]	To get/set the active simulation data source.
DataStreamNames	DataStreamNames[2)]	To get information of the data stream.
IsAnimationRunning	Boolean	To get the state of the animation: ▪ False: Animation is stopped. ▪ True: Animation is running.
MaxSimulationFrames	Integer	To get/set the maximum number of simulation frames.
SimulationBuffer	SimulationBufferManager[3)]	To get information on the simulation buffer.

Attributes	Type	Purpose
SimulationDataPointADAS	SimulationDataPointADAS[4]	To specify an ADAS Simulator to be used as the data source for motion data.
SimulationDataPointFile	SimulationDataPointFile[5]	To specify an MDF file to be used as the data source for motion data.
SimulationDataPointHardware	SimulationDataPointHardware[6]	To specify a dSPACE hardware platform connected via bus connection to be used as the data source for motion data.
SimulationDataPointHardwareNet	SimulationDataPointHardwareNet[7]	To specify a dSPACE hardware platform connected via network connection to be used as the data source for motion data.
SimulationDataPointMlbx	SimulationDataPointMlbx[8]	To specify a MicroLabBox platform to be used as the data source for motion data
SimulationDataPointMABX	SimulationDataPointMABX[9]	To specify a MicroAutoBox III platform to be used as the data source for motion data
SimulationDataPointNetwork	SimulationDataPointNetwork[10]	To specify a network adapter to be used as the data source for motion data.
SimulationDataPointSCALEXIO	SimulationDataPointScalexio[11]	To specify a SCALEXIO platform to be used as the data source for motion data.
SimulationDataPointVEOS	SimulationDataPointVEOS[12]	To specify a VEOS platform to be used as the data source for motion data.
SimulationDataPointDS1007	SimulationDataPointDS1007[13]	To specify a DS1007 platform to be used as the data source for motion data.

1) Refer to SimulationDataSources on page 277.
2) Refer to DataStreamNames on page 222.
3) Refer to SimulationBufferManager on page 237.
4) Refer to SimulationDataPointADAS on page 241.
5) Refer to SimulationDataPointFile on page 242.
6) Refer to SimulationDataPointHardware on page 242.
7) Refer to SimulationDataPointHardwareNet on page 245.
8) Refer to SimulationDataPointMlbx on page 247.
9) Refer to SimulationDataPointMABX on page 251.
10) Refer to SimulationDataPointNetwork on page 255.
11) Refer to SimulationDataPointScalexio on page 256.
12) Refer to SimulationDataPointVEOS on page 260.
13) Refer to SimulationDataPointDS1007 on page 264.

Methods

The class contains the following methods:

Method	Purpose
StartAnimation	To start the animation. Refer to StartAnimation on page 271.
StopAnimation	To stop the animation. Refer to StopAnimation on page 271.

Related topics

References

Class Description (Application) (MotionDesk Basics 🕮)

StartAnimation

Class

SimulationManager

Syntax

```
MySimulationManagement.StartAnimation()
```

Purpose

To start the animation.

Parameters

–

Return value

The method returns the following parameter:

Type	Description
Boolean	True if the animation is running.

Related topics

References

Class Description (SimulationManager)..... 269

StopAnimation

Class

SimulationManager

Syntax

```
MySimulationManagement.StopAnimation()
```

Purpose

To stop the animation.

Parameters

–

Return value

The method returns the following parameter:

Type	Description
Boolean	True if the animation is stopped.

Related topics

References

Class Description (SimulationManager)..... 269

SimulationMotionData

Purpose

To get the numerical values of the motion data.

Class Description (SimulationMotionData)

Syntax

```
SimulationManager = Application.SimulationManagement
SimulationBuffer = SimulationManager.SimulationBuffer
FrameData = SimulationBuffer.FrameData
FrameData.MotionData(0)
```

Purpose

To get the numerical values of the motion data.

Attributes

The class contains the following attributes:

Attributes	Type	Purpose
Rotation	Float[]	To get the values of the rotation matrix.
Translation	Float[]	To get the values of the translation matrix.

Methods

–

Related topics

References

Class Description (Application) (MotionDesk Basics 🕮)
Class Description (SimulationManager)..... 269

Snow

Purpose

To get or set the settings of snow scenarios.

Class Description (Snow)

Syntax

```
Snow = CustomAtmosphericsSettings.Snow
```

Purpose

To get or set the settings of snow scenarios.

Attributes

The class contains the following attributes:

Attributes	Type	Purpose
Enabled	Boolean	To get/set the switch for enabling the snow.
Color	Integer[]	To get/set the color as a tuple of 3 integer value (RGB).
Speed	Double	To get/set the speed.
Intensity	Double	To get/set the intensity.
ParticelSize	Double	To get/set the particle size.
FallingAngle	Double	To get/set the falling angle.
HeadingAngle	Double	To get/set the heading angle.

Methods

–

Related topics

References

Atmospherics Properties..... 189

Sunlight

Purpose

To get or set the settings of sunlight.

Class Description (Sunlight)

Syntax

```
Sunlight = CustomAtmosphericsSettings.Sunlight
```

Purpose

To get or set the settings of sunlight.

Attributes

The class contains the following attributes:

Attributes	Type	Purpose
Enabled	Boolean	To get/set the switch for enabling the sunlight.
Intensity	Double	To get/set the intensity.
Color	Integer[]	To get/set the color as a tuple of 3 integer value (RGB).
ElevationAngle	Boolean	To get/set the elevation angle.
AzimuthAngle	Boolean	To get/set the azimuth angle.

Methods

–

Related topics

References

Atmospherics Properties..... 189
Class Description (CustomAtmospherics)..... 221

VisualizationManager

Purpose

To access movable objects, static objects, atmospherics, sensors, and views of a scene.

Class Description (VisualizationManager)

Syntax

```
VisualizationManager =
Experiment.VisualizationManagement
```

Purpose

To access movable objects, static objects, atmospherics, sensors, and views of a scene.

Attributes

The class contains the following attributes:

Attributes	Type	Purpose
Atmospherics	AtmosphericsManager[1)]	To access the atmospherics.
FramesPerSecond	Double	To get the frame rate in frames per second.
MovableObjects	MovableObjectManager[2)]	To access movable objects.
Observers	ObserverManager[3)]	To access the observers.
Sensors	Sensors[4)]	To access the sensors in a MotionDesk scene.
StaticObjects	StaticObjectManager[5)]	To access static objects.
Views	ViewManager[6)]	To access the views.

1) Refer to AtmosphericsManager on page 220.
2) Refer to MovableObjectManager (MotionDesk Scene Creation 🕮).
3) Refer to ObserverManager on page 229.
4) Refer to Sensors (MotionDesk Sensor Simulation Control 🕮).
5) Refer to StaticObjectManager (MotionDesk Scene Creation 🕮).
6) Refer to ViewManager (MotionDesk Basics 🕮).

Methods

–

Related topics

References

Class Description (Experiment) (MotionDesk Project and Experiment Management 🕮)

Enumerations

Enumerations for Scene Animation

Introduction

You can use predefined constants in the tool automation.

Enumerations

The following constants can be used for tool automation.

AtmosphericsModes Constants to specify the atmospherics mode.

Value	Description
Sunny	A predefined sunny atmosphere
Hazy	A predefined hazy atmosphere
Foggy	A predefined foggy atmosphere
Sunset	A predefined sunset atmosphere
Snowy	A predefined snowy atmosphere
Rainy	A predefined rainy atmosphere
Night	A predefined night atmosphere
Custom	A custom defined atmosphere

AtmosphericPresetModes Constants to specify the preset for a custom atmospheric setting.

Value	Description
Sunny	A preset for a sunny atmosphere
Hazy	A preset for a hazy atmosphere
Foggy	A preset for a foggy atmosphere
Sunset	A preset for a sunset atmosphere
Night	A preset for a night atmosphere
Rainy	A preset for a rainy atmosphere
Snowy	A preset for a snowy atmosphere

DensityModes Constants to specify the density mode:

Value	Description
Exponential	The fog increases exponentially.
ExpotentialSquared	The fog increases exponentially squared.

FollowBehaviour Constants to specify the behavior of an observer.

Value	Description
Free	Observer can be moved freely in the virtual world.
FixedIntertial	Observer is fixed to the virtual world.

Value	Description
FixedToObject	Observer is fixed to a movable object.
FollowDriver	Observer follows the attached movable object. The view is from the driver's seat.
FollowDriverSmooth	Observer follows the attached movable object smoothly. The view is from the driver's seat.
FollowRotation	Observer follows the rotation of the attached movable object.
FollowTranslation	Observer follows the translation of the attached movable object.
FollowXTranslation	Observer follows the x translation of the attached movable object.
FollowYTranslation	Observer follows the y translation of the attached movable object.

IdentifierTypes Constants to identify objects.

Value	Description
ByName	Object is identified by its name.
ByIndex	Object is identified by its index.
ByCustomIdentifier	Object is identified by a custom identifier.

PHSHardware Constants to specify a dSPACE platform.

Value	Description
Undefined	Platform is not defined.
DS1006	Platform is a DS1006.
MABX	Platform is MicroAutoBox.
DS1007	Platform is a DS1007.

SimulationDataSources Constants to specify the data source for motion data.

Value	Description
None	No data source specified.
ADAS	The data source is an ADAS simulator.
File	The data source is an MDF file.
Hardware	The data source is a dSPACE hardware connected via bus connection.
HardwareNet	The data source is a dSPACE hardware connected via network connection.
Network	The data source is connected via network adapter.
SCALEXIO	The data source is a SCALEXIO system.
VEOS	The data source is a VEOS.
DS1007	The data source is a DS1007.
MLBX	The data source is a MicroLabBox.
MABX	The data source is a MicroAutoBox III.

ViewModes Constants to specify the view mode:

Value	Description
OneView	The scene is displayed in one view.
Horizontal	The scene is displayed in two views, horizontally arranged.

Value	Description
Vertical	The scene is displayed in two views, vertically arranged.
FourViews	The scene is displayed in four views.

Related topics

References

Enumerations for Handling MotionDesk (MotionDesk Basics 🕮)
Enumerations for Scene Creation (MotionDesk Scene Creation 🕮)

Limitations

Limitations of Scene Animation

Introduction	The following limitations apply for scene animation.
Data acquisition	The time stamps in MDF files generated before MotionDesk 2.0 are not evaluated. The animation runs as fast as possible and is not synchronized to time.
Limitations when working with tracks	▪ MDF files recorded in RealMotion cannot be used with tracks. MotionDesk will generate an error message. ▪ Track-safety requires lossless data acquisition and maneuver synchronization. For MDF files recorded before MotionDesk 2.0, only lossless data acquisition is available but no maneuver synchronization. Therefore, these MDF files are not track safe and MotionDesk will generate a warning message in the **Message Viewer**. ▪ If you deleted a movable object of a track by mistake, you can restore it. Drag it from its original in the **Movable Objects** folder. You can create only a default clone. All previous modifications are lost.
Limited frame buffer size	When a real-time application is running, motion data is stored in a frame buffer first. The buffer size is limited to 30,000 frames. For example, if 150 movable objects are used, a memory of about 500 MB is necessary.
Limitations of observers	An observer with the "follow driver" behavior does not follow the car's pitch angle. Thus, in automotive applications with a z-profiled road, you will look at the pedals if you drive up a slope.

Limitations when using instruments

The multi-track mode is not supported for instruments in MotionDesk.

MotionDesk's instruments do not support script fonts, such as the *Vladimir Script* font. You can identify script fonts by their name.

Troubleshooting (Scene Animation)

Where to go from here

Information in this section

Information in other sections

Troubleshooting (Scene Creation) (MotionDesk Scene Creation 🕮)

Troubleshooting (General) (MotionDesk Basics 🕮)

Other Programs Run Slowly When MotionDesk Runs

Problem

When MotionDesk runs, other programs run slowly.

Reason

When MotionDesk runs, it captures all the processor time which is not needed by other programs. If another running program requires more processor time, MotionDesk must free some processor time first.

Solution

- Minimize MotionDesk.

  When MotionDesk is minimized, the animation is stopped. The other programs can get more processor time.
- Stop the animation.

  When the animation is stopped, MotionDesk does not acquire motion data from the data source and the processor load is therefore reduced.
- Reduce the maximum frame rate.

  You can use a lower frame rate for the animation. For details, refer to Visualization Options Page on page 214.

Starting Other Programs is Slow

Problem

When MotionDesk is running, it takes longer to start other programs.

Reason

When MotionDesk runs, it captures all the available processor time for smooth visualization. When other programs are started, MotionDesk must first free some of the resources used.

Solution

- Start all other programs before starting MotionDesk.
- Minimize MotionDesk.

Some Objects Do Not Move During Animation

Problem	The animation is started but some movable objects do not move.
Reason	Motion data is not assigned to these objects.
Solution	Assign a data stream to the objects. Refer to How to Assign Motion Data to Movable Objects on page 26.

An Instrument Displays the Value '0' During Animation

Problem	The animation is started but instruments displays only value '0'.
Reason	The instruments are not connected to an appropriate data stream. The instrument icon in the **Scene Navigator** has a question mark or exclamation mark.
Solution	Select an appropriate data stream in the **Signal data** property of the instrument.

No Objects Move During Animation

Problem	The animation is started but no object moves.
Reason	The data source is not selected or is incorrectly configured.
Solution	Select a data source correctly. Refer to How to Select the Data Source on page 16.

Animation is not Smooth

Problem	The animation is not smooth or the frame rate is low.
Reason	Your visualization PC is too slow or your scene has too many polygons or textures.
Solution	Refer to Getting Smooth Visualization on page 123.

Animation Freezes During an Online Simulation

Problem	When playing an online vehicle simulation in MotionDesk, the scene animation freezes.
Reason	This can occur when you connect to a simulation platform using a TCP connection. The scene animation can freeze after you select a different observer view using the animation pane context menu. The online simulation TCP connection can timeout when the main thread is blocked by another process.
Solution	You must go offline to disconnect the TCP connection to the simulation and then go online to reconnect and resume the animation. 1. Select **Go Offline** in the **Simulation** ribbon. 2. Select **Go Online** in the **Simulation** ribbon.

Fellows Are Not Displayed in MotionDesk After Some Time

Problem	MotionDesk visualizes a traffic scenario that has several fellows, for example, 15 or more. After some time, the fellows are no longer displayed.
Reason	Working with several fellows requires a large amount of memory of the graphics adapter. The problem occurs especially if the graphics adapter has only 1 GB of memory or less.

Solution	Before loading the MotionDesk project, increase the texture compression. Open the **Render Options** page and specify **Maximum compression** for the texture compression. Refer to Render Options Page on page 208.

MotionDesk PC Crashes

Problem	The MotionDesk PC crashes or boots automatically after a few minutes.
Reason	The graphics driver may be erroneous.
Solution	Install the latest graphics driver for your graphics card. If your graphics card has the NVIDIA chip (GPU), you can use the NVIDIA reference driver.

MotionDesk PC Cannot Connect to Data Source via Ethernet

Problem	The MotionDesk PC has two or more network adapters and cannot connect to the simulator via Ethernet.
Reason	If a MotionDesk PC has several network adapters, it can happen that the operating system does not use the configured network adapter to connect to the simulator.
Solution	Specify the IP address of the simulator. Refer to Customize Data Source on page 172.

When Comparing a Simulation With Recorded Tracks the Animation is Jerky

Problem	When comparing a simulation with a track, the track animation is jerky.
Reason	If you open a dialog, this stops the recording of the simulation buffer as long as the dialog is open. This causes jumps in the animation.

Solution	Do not open any dialogs when recording the simulation buffer for a track.

Numerics

A

B

C

D

E

F

G

H

I

L

M

N

O