EcoStruxure Machine Expert Twin

Getting Started

User Guide

EIO0000005022.04
12/2024

www.se.com

Legal Information

The information provided in this document contains general descriptions, technical characteristics and/or recommendations related to products/solutions.

This document is not intended as a substitute for a detailed study or operational and site-specific development or schematic plan. It is not to be used for determining suitability or reliability of the products/solutions for specific user applications. It is the duty of any such user to perform or have any professional expert of its choice (integrator, specifier or the like) perform the appropriate and comprehensive risk analysis, evaluation and testing of the products/solutions with respect to the relevant specific application or use thereof.

The Schneider Electric brand and any trademarks of Schneider Electric SE and its subsidiaries referred to in this document are the property of Schneider Electric SE or its subsidiaries. All other brands may be trademarks of their respective owner.

Schneider Electric does not grant any right or license for commercial use of the document or its content, except for a non-exclusive and personal license to consult it on an "as is" basis.

Schneider Electric reserves the right to make changes or updates with respect to or in the content of this document or the format thereof, at any time without notice.

To the extent permitted by applicable law, no responsibility or liability is assumed by Schneider Electric and its subsidiaries for any errors or omissions in the informational content of this document, as well as any non-intended use or misuse of the content thereof.

Table of Contents

Safety Information

Important Information

Read these instructions carefully, and look at the equipment to become familiar with the device before trying to install, operate, service, or maintain it. The following special messages may appear throughout this documentation or on the equipment to warn of potential hazards or to call attention to information that clarifies or simplifies a procedure.

The addition of this symbol to a “Danger” or “Warning” safety label indicates that an electrical hazard exists which will result in personal injury if the instructions are not followed.

This is the safety alert symbol. It is used to alert you to potential personal injury hazards. Obey all safety messages that follow this symbol to avoid possible injury or death.

⚠ DANGER

DANGER indicates a hazardous situation which, if not avoided, **will result in** death or serious injury.

⚠ WARNING

WARNING indicates a hazardous situation which, if not avoided, **could result in** death or serious injury.

⚠ CAUTION

CAUTION indicates a hazardous situation which, if not avoided, **could result** in minor or moderate injury.

NOTICE

NOTICE is used to address practices not related to physical injury.

Please Note

Electrical equipment should be installed, operated, serviced, and maintained only by qualified personnel. No responsibility is assumed by Schneider Electric for any consequences arising out of the use of this material.

A qualified person is one who has skills and knowledge related to the construction and operation of electrical equipment and its installation, and has received safety training to recognize and avoid the hazards involved.

About the Book

Document Scope

This document describes the graphical user interface of the EcoStruxure Machine Expert Twin software and the functions it provides. For further information, refer to the separate documents provided in the EcoStruxure Machine Expert Twin online help.

Validity Note

This document has been updated for the release of EcoStruxure Machine Expert Twin V2.2.

Related Documents

Document title	Reference
Cybersecurity Best Practices	CS-Best-Practices-2019-340
Cybersecurity Guidelines for EcoStruxure Machine Expert, Modicon and PacDrive Controllers and Associated Equipment	EIO0000004242
EcoStruxure Automation Expert - Motion, EcoStruxure Machine Expert DigitalTwinCommunication Library Guide	EIO0000004735 (ENG)
EcoStruxure Machine Expert Twin How to Create Device Catalogs - User Guide	EIO0000005034 (ENG)

To find documents online, visit the Schneider Electric download center (www.se.com/ww/en/download/).

Product Related Information

⚠WARNING

LOSS OF CONTROL

- Perform a Failure Mode and Effects Analysis (FMEA), or equivalent risk analysis, of your application, and apply preventive and detective controls before implementation.
- Provide a fallback state for undesired control events or sequences.
- Provide separate or redundant control paths wherever required.
- Supply appropriate parameters, particularly for limits.
- Review the implications of transmission delays and take actions to mitigate them.
- Review the implications of communication link interruptions and take actions to mitigate them.
- Provide independent paths for control functions (for example, emergency stop, over-limit conditions, and error conditions) according to your risk assessment, and applicable codes and regulations.
- Apply local accident prevention and safety regulations and guidelines.[1]
- Test each implementation of a system for proper operation before placing it into service.

Failure to follow these instructions can result in death, serious injury, or equipment damage.

[1] For additional information, refer to NEMA ICS 1.1 (latest edition), *Safety Guidelines for the Application, Installation, and Maintenance of Solid State Control* and to NEMA ICS 7.1 (latest edition), *Safety Standards for Construction and Guide for Selection, Installation and Operation of Adjustable-Speed Drive Systems* or their equivalent governing your particular location.

⚠WARNING

UNINTENDED EQUIPMENT OPERATION

- Only use software approved by Schneider Electric for use with this equipment.
- Update your application program every time you change the physical hardware configuration.

Failure to follow these instructions can result in death, serious injury, or equipment damage.

For reasons of Internet security, for those devices that have a native Ethernet connection, TCP/IP forwarding is disabled by default. Therefore, you must manually enable TCP/IP forwarding. However, doing so may expose your network to possible cyberattacks if you do not take additional measures to protect your enterprise. In addition, you may be subject to laws and regulations concerning cybersecurity.

⚠WARNING

UNAUTHENTICATED ACCESS AND SUBSEQUENT NETWORK INTRUSION

- Observe and respect any and all pertinent national, regional and local cybersecurity and/or personal data laws and regulations when enabling TCP/IP forwarding on an industrial network.
- Isolate your industrial network from other networks inside your company.
- Protect any network against unintended access by using firewalls, VPN, or other, proven security measures.

Failure to follow these instructions can result in death, serious injury, or equipment damage.

Consult the Schneider Electric Cybersecurity Best Practices for additional information.

EcoStruxure Machine Expert Twin is a simulation and emulation software suite to create digital models of real machines to start the virtual design, virtual pre-commissioning, and to support co-development before building the machine – thus enabling parallel engineering of mechanical, electrical and controls work assignments.

The simulation, emulation and machine visualization functions of EcoStruxure Machine Expert Twin are intended to support you in developing your application and its configuration by simulating the behavior of the various machine or process components. These functions are not intended to substitute for, but to complement the processes of risk assessment, risk evaluation, validation, and commissioning as well as any ancillary processes, tasks, and obligations according to the applicable regulations and standards such as ISO/EN 13849 and IEC 62061. The product, though powerful, does not, nor can it, simulate every aspect of the application and its environment.

⚠WARNING

INSUFFICIENT TEST COVERAGE

- Do not use EcoStruxure Machine Expert Twin as the sole means for risk assessment, risk evaluation, validation, and commissioning as well as any ancillary processes, tasks, and obligations according to the applicable regulations and standards such as, but not limited to, ISO/EN 13849 and IEC 62061.
- Verify and validate your results on the intended equipment before placing your machine or process into service.

Failure to follow these instructions can result in death, serious injury, or equipment damage.

Based on the system configuration and operation, a hazard and risk analysis must be conducted for the system (for example, according to ISO 12100 or ISO 13849-1) independent of the work with EcoStruxure Machine Expert Twin. The results of this analysis must be considered when designing the machine, and subsequently applying safety-related equipment and safety-related functions. The results of your analysis may deviate from any digital models of physical machines that you may create. For example, additional safety components may be required. In principle, the results from the hazard and risk analysis have priority.

⚠WARNING

NON-CONFORMANCE TO SAFETY FUNCTION REQUIREMENTS

- Specify the requirements and/or measures to be implemented in the risk analysis you perform.
- Verify that your safety-related application complies to applicable safety regulations and standards.
- Make certain that appropriate procedures and measures (according to applicable sector standards) have been established to help avoid hazardous situations when operating the machine.
- Use appropriate safety interlocks where personnel and/or equipment hazards exist.
- Validate the overall safety-related function and thoroughly test the application.

Failure to follow these instructions can result in death, serious injury, or equipment damage.

Catalogs contain important data, parameters and operational aspects of the devices defined within. This information is subject to change over time for a variety of reasons. Therefore, it is necessary to maintain the relationship between the models you create and the catalogs you have used to do so. Version mismatches of catalogs may cause your models to operate in ways that are incongruent with the equipment they represent and may lead to errors in design and operation.

⚠WARNING

UNINTENDED EQUIPMENT OPERATION

- Impose a system of file name conventions that readily indicate the version of the catalogs you use and models you create.
- Create documentation that records catalog and model versions, as well as firmware versions of the equipment used in your models.

Failure to follow these instructions can result in death, serious injury, or equipment damage.

In addition, changes to your underlying application (logic, data address, functions, I/O configurations, device types and configuration, etc.) can have serious impact on the models you have created.

⚠WARNING

UNINTENDED EQUIPMENT OPERATION

- Update your models every time you modify your application or change the physical hardware configuration.
- Verify that objects you have created in your models are coherent with the modifications and/or changes you have made to your application and that they are associated with the correct variables.

Failure to follow these instructions can result in death, serious injury, or equipment damage.

It is also important to connect to the correct automation logic/motion controller in a networked, multi-controller environment.

⚠ WARNING
UNINTENDED EQUIPMENT OPERATION Verify that you have connected to the intended automation controller. **Failure to follow these instructions can result in death, serious injury, or equipment damage.**

It is important to manage the amount of data that is transmitted between your automation logic/motion controller and EcoStruxure Machine Expert Twin. Large amounts of data, or data that is not contiguous in the controller memory may impact performance of EcoStruxure Machine Expert Twin, the controller or both.

Information on Non-Inclusive or Insensitive Terminology

As a responsible, inclusive company, Schneider Electric is constantly updating its communications and products that contain non-inclusive or insensitive terminology. However, despite these efforts, our content may still contain terms that are deemed inappropriate by some customers.

Terminology Derived from Standards

The technical terms, terminology, symbols and the corresponding descriptions in the information contained herein, or that appear in or on the products themselves, are generally derived from the terms or definitions of international standards.

In the area of functional safety systems, drives and general automation, this may include, but is not limited to, terms such as *safety*, *safety function*, *safe state*, *fault*, *fault reset*, *malfunction*, *failure*, *error*, *error message*, *dangerous*, etc.

Among others, these standards include:

Standard	Description
IEC 61131-2:2007	Programmable controllers, part 2: Equipment requirements and tests.
ISO 13849-1:2023	Safety of machinery: Safety related parts of control systems. General principles for design.
EN 61496-1:2020	Safety of machinery: Electro-sensitive protective equipment. Part 1: General requirements and tests.
ISO 12100:2010	Safety of machinery - General principles for design - Risk assessment and risk reduction
EN 60204-1:2006	Safety of machinery - Electrical equipment of machines - Part 1: General requirements
ISO 14119:2013	Safety of machinery - Interlocking devices associated with guards - Principles for design and selection
ISO 13850:2015	Safety of machinery - Emergency stop - Principles for design
IEC 62061:2021	Safety of machinery - Functional safety of safety-related electrical, electronic, and electronic programmable control systems
IEC 61508-1:2010	Functional safety of electrical/electronic/programmable electronic safety-related systems: General requirements.
IEC 61508-2:2010	Functional safety of electrical/electronic/programmable electronic safety-related systems: Requirements for electrical/electronic/programmable electronic safety-related systems.

Standard	Description
IEC 61508-3:2010	Functional safety of electrical/electronic/programmable electronic safety-related systems: Software requirements.
IEC 61784-3:2021	Industrial communication networks - Profiles - Part 3: Functional safety fieldbuses - General rules and profile definitions.
2006/42/EC	Machinery Directive
2014/30/EU	Electromagnetic Compatibility Directive
2014/35/EU	Low Voltage Directive

In addition, terms used in the present document may tangentially be used as they are derived from other standards such as:

Standard	Description
IEC 60034 series	Rotating electrical machines
IEC 61800 series	Adjustable speed electrical power drive systems
IEC 61158 series	Digital data communications for measurement and control – Fieldbus for use in industrial control systems

Finally, the term *zone of operation* may be used in conjunction with the description of specific hazards, and is defined as it is for a *hazard zone* or *danger zone* in the *Machinery Directive* (*2006/42/EC*) and *ISO 12100:2010*.

NOTE: The aforementioned standards may or may not apply to the specific products cited in the present documentation. For more information concerning the individual standards applicable to the products described herein, see the characteristics tables for those product references.

Introduction

What’s in This Part

General Introduction to the EcoStruxure Machine Expert Twin

What's in This Chapter

Overview of the EcoStruxure Machine Expert Twin Product Portfolio

Introduction

The EcoStruxure Machine Expert Twin product portfolio consists of the following segments:

- EcoStruxure Automation Expert - Motion / EcoStruxure Machine Expert Visu

  This viewer integrated in EcoStruxure Automation Expert - Motion and EcoStruxure Machine Expert provides a subset of the functions of EcoStruxure Machine Expert Twin. It allows you to test the application code and to display a 3-D emulation of the objects, for example, one multi carrier track or one robot, from within EcoStruxure Automation Expert - Motion and EcoStruxure Machine Expert. You can reuse the scenes of single machines to virtual commissioning of the entire machine production line inside EcoStruxure Machine Expert Twin Builder.
- EcoStruxure Machine Expert Twin

  The standalone EcoStruxure Machine Expert Twin application is installed using the Schneider Electric Software Installer and requires a separate license. It includes different components:
  - EcoStruxure Machine Expert Twin Builder

    The Builder License allows the emulation of a machine when the controller is connected. It includes predefined catalogs that provide objects you can use as assemblies in your scene.
  - EcoStruxure Machine Expert Twin Designer

    The EcoStruxure Machine Expert Twin Designer allows the emulation of machines without a controller being connected (discrete events simulation).
  - EcoStruxure Machine Expert Twin Developer

    The Developer License allows you to create your own emulation components and catalogs as well as plugins, tools, interfaces you can adapt to your workflow, processes and data sources.

User Interface

What's in This Part

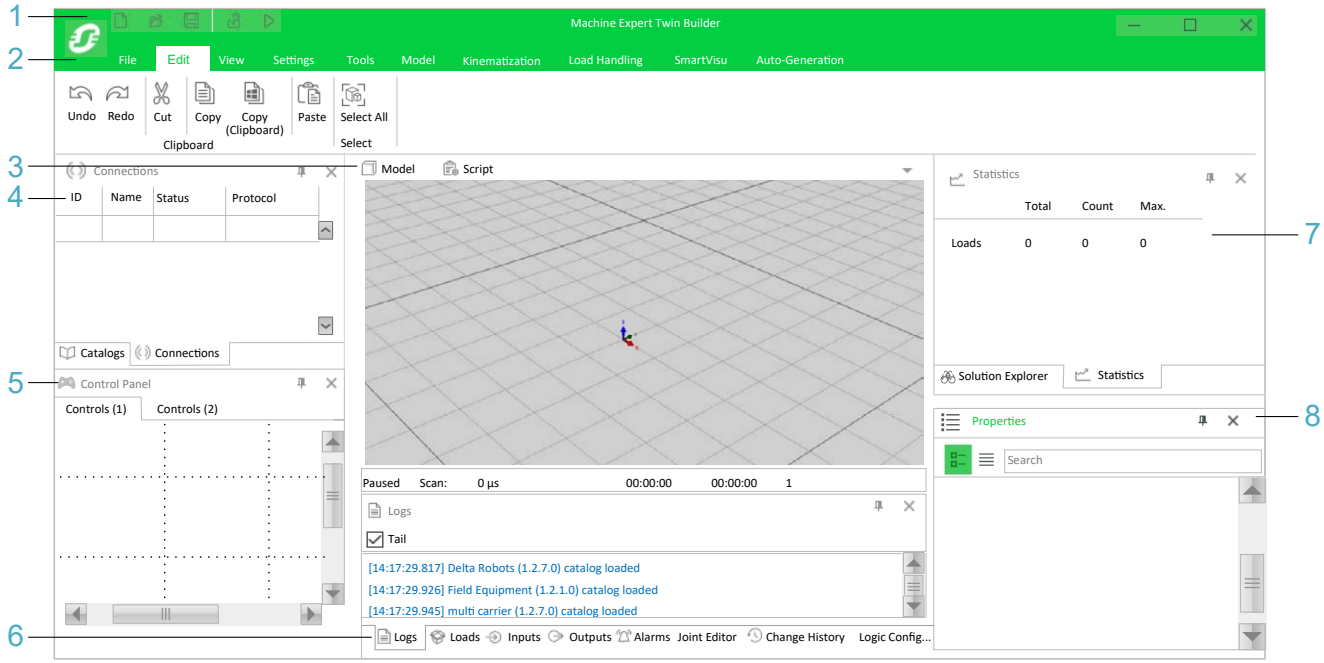
Overview of the User Interface

What's in This Chapter

Elements of the Screen

Default Screen

After initial startup, the default screen is displayed. It consists of the following elements:



1 Toolbar, page 20

2 Menu bar, page 21 with button bar representing the commands corresponding to the selected menu

3 **Model** view, page 72, **Script** view, page 82

4 **Connections** view, page 67 / **Catalogs** view, page 69

5 **Control Panel** view, page 70 providing tabs for different controls

6 **Logs** view, page 83, **Loads** view, page 85, **Inputs**, **Outputs** views, page 86, **Alarms** view, page 87, **Joint Editor** view, page 88, **Logic Configurator** view, page 92, **Change History** view, page 93

7 **Statistics** view, page 97 / **Solution Explorer** view, page 94

8 **Properties** view, page 98

Information About the Coordinate System

General Definition for 3-D Software Applications

In 3-D software applications, a coordinate system is a mathematical system that is used to represent the position and orientation of objects in a three-dimensional space. It consists of three perpendicular axes (X axis, Y axis, and Z axis) that intersect at a point called the origin.

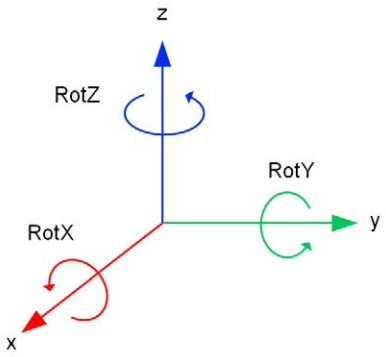
Color Definition of Coordinates and Denomination of Rotation

The axes of the coordinate system in EcoStruxure Machine Expert Twin are color-coded as follows:

- X: red
- Y: green
- Z: blue

Denomination of rotational movement around the different axes is as follows:

- Rotation around the X axis: **RotX**
- Rotation around the Y axis: **RotY**
- Rotation around the Z axis: **RotZ**



Intrinsic Rotations Convention

With reference to the global EcoStruxure Machine Expert Twin coordinate system, rotations of local assembly coordinate systems of individual body assemblies are performed according to the intrinsic convention with the default orientation convention ZYX. In an intrinsic system, each of the elemental rotations is performed on the coordinate system as rotated by the previous operation(s).

As an example, suppose the three angles specify rotations around the Z, Y, and X axes are in that order:

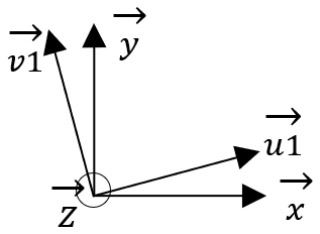
- The first elemental rotation is around the Z axis.
- For the intrinsic convention the second elemental rotation is performed around the Y axis in the new position resulting from the first rotation.
- The final rotation around the X axis is performed around the X axis as rotated by the two previous operations in the intrinsic system.

The relationship between the three axes is fundamental in EcoStruxure Machine Expert Twin and is used to determine the orientation and position of body assemblies in a scene.

Order of Rotational Movements

When values for rotational movements of the different types are provided at the same time, for example, if an object is added by using the orientation values from the OPC UA structures, rotational movements are executed in the following order:

1. **Around the Z axis:**

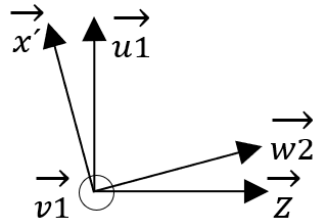


The vector *z* is pointing in your direction.

$$rotZ = (\vec{x}, \overrightarrow{u_1}) = (\vec{y}, \overrightarrow{v_1})$$

The intermediate vectors *u1* and *v1* are implicit.

2. **Around the Y axis:**

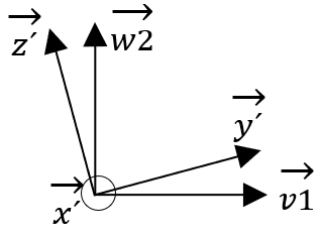


The vector v_1 is pointing in your direction.

$$rotY = (\vec{z}, \overrightarrow{w_2}) = (\overrightarrow{u_1}, \overrightarrow{x'})$$

The intermediate vectors *u1*, *v1*, and *w2* are implicit.

3. **Around the X axis:**



The vector *x'* is pointing in your direction.

$$rotX = (\overrightarrow{v_1}, \overrightarrow{y'}) = (\overrightarrow{w_2}, \overrightarrow{z'})$$

The intermediate vectors *u1*, *v1*, and *w2* are implicit.

Toolbar

What's in This Chapter

Toolbar

Default Elements of the Toolbar

By default, the toolbar provides the following elements:

Element	Shortcut	Description
New	**Ctrl** + **N**	Create a new project.
Open	**Ctrl** + **O**	Open an existing project. Your file contains script code created or modified by you or by someone else. The execution of the script code may compromise the security of your IT system. **⚠WARNING** **INFORMATION SYSTEM VULNERABILITY** • Ensure the integrity of the script code contained in the file by ascertaining the originator and intent of the script before opening the file. • Do not open the file if you cannot determine the originator and intent of the script, including confirming any modifications that may have been made. • Verify and confirm that you want to execute this code when using the model contained in the file. **Failure to follow these instructions can result in death, serious injury, or equipment damage.** When attempting to open a file with a model containing script code, you are asked to acknowledge this advisory and accept responsibility for the execution of the code of the embedded script.
Save	**Ctrl** + **S**	Save the open project with file extension *.Experior*.
Lock	**Ctrl** + **L**	Lock the open project: the scene is locked against edits, no modifications are possible, values are greyed and buttons are unavailable. To lock specific elements of a scene, such as an assembly or a section, refer to the **Solution Explorer** view, page 94 or the **Properties**, page 98 of the specific element.
Play	-	Play the scene. The **Play** button enables the physical simulation.
Show Quick Access Toolbar Below the Ribbon / **Show Quick Access Toolbar Above the Ribbon**	-	Click **Show Quick Access Toolbar Below the Ribbon** to shift the toolbar below the menu bar. Click **Show Quick Access Toolbar Above the Ribbon** to shift the toolbar above the menu bar. Right-click this toggle buttons to access the following commands: • **Minimize the Ribbon**: Run this command from the contextual menu to display or hide the button bar corresponding to the menu selected in the menu bar: The buttons are hidden until you run the **Minimize the Ribbon** command again or until you select another menu in the menu bar. • **Customize the Ribbon** Opens the **Ribbon customization window** and allows you to customize the buttons and commands displayed in the menus.

Menu Bar

What's in This Chapter

The menu bar is by default extended by a button bar that provides the commands of each menu in the form of buttons. This chapter describes the default commands, provided by each menu.

File Menu

Commands of the File Menu

The **File** menu is grouped in different sections:

File Section

Command	Shortcut	Description
New	**Ctrl** + **N**	Creates a new project. Your file contains script code created or modified by you or by someone else. The execution of the script code may compromise the security of your IT system. **⚠WARNING** **INFORMATION SYSTEM VULNERABILITY** • Ensure the integrity of the script code contained in the file by ascertaining the originator and intent of the script before opening the file. • Do not open the file if you cannot determine the originator and intent of the script, including confirming any modifications that may have been made. • Verify and confirm that you want to execute this code when using the model contained in the file. **Failure to follow these instructions can result in death, serious injury, or equipment damage.** When attempting to open a file with a model containing script code, you are asked to acknowledge this advisory and accept responsibility for the execution of the code of the embedded script.
Open	**Ctrl** + **O**	Opens an existing project. As a project file contains executable script code, carefully verify whether this project file originates from a trusted source because opening a tampered project file can compromise the security of your IT system before you click **Exit** or activate the check box and click **Continue**.
Save	**Ctrl** + **S**	Saves the open project with file extension *.Experior*.
Save As	**Ctrl** + **Shift** + **S**	Saves the open project with a file name and in the directory of your choice.

Import Section

Command	Shortcut	Description
Merge	-	Allows you to merge two parts of a machine that have been designed independently as scenes and saved as EcoStruxure Machine Expert Twin projects in *.Experior* file format. For a step-by-step description, refer to *Merging Projects*, page 23.
XML	-	Imports an XML file that has an EcoStruxure Machine Expert Twin-compatible format. Files of type *.Experior* or *.zip* are supported.

Also refer to Importing a CAD File, page 35 in the **Kinematization** Menu, page 31.

Merging Projects

To merge two parts of a machine that have been designed as independent *.Experior* projects, proceed as follows:

Step	Action
1	Create an EcoStruxure Machine Expert Twin project in *.Experior* file format for a part of your machine, for example, a conveyor belt.
2	Save and close this project file.
3	Create a second EcoStruxure Machine Expert Twin project in *.Experior* file format and design another part of your machine, for example, a robot.
4	Save the second file and click **File > Merge** to display a file open dialog box.
5	Browse to the *.Experior* file you created for the conveyor belt and click **Open**. **Result**: The **Merge/Overwrite Objects** dialog box opens.
6	Verify the objects that are selected in the dialog box for **Insert** and **Overwrite** and adapt the selections to your needs.
7	Select the option **Reuse sections (if they exist in the current model)** to add the assemblies of the conveyor belt project to be merged to an existing section of the open robot project if the section name is identical and click **OK**. **Result**: In the **Solution Explorer** view, the conveyor belt is added to the respective **Section** node.

For a visual illustration of this merging process, refer to the video sequence in the corresponding online help.

Export Section

Command	Shortcut	Description
Collada	-	Exports the assemblies in the scene as a Collada (COLLAborative Design Activity) file with the file extension *.dae* to allow for exchange with other applications.

For importing a Collada file, refer to Importing a CAD File, page 35 in the **Kinematization** Menu, page 31.

Help Section

Command	Shortcut	Description
About	-	Opens a dialog box providing information about the components (such as catalogs or plugins) that are loaded. Furthermore, it provides links to the EcoStruxure Machine Expert Twin Release Notes and to the Schneider Electric License Manager allowing you to verify the license status.
Help	-	Opens the EcoStruxure Machine Expert Twin Online Help.

Edit Menu

Commands of the Edit Menu

The **Edit** menu is grouped in different sections:

Clipboard Section

Command	Shortcut	Description
Undo	**Ctrl** + **Z**	Undoes the action which was most recently executed. Repeated use undoes the actions back to when the project was opened.
Redo	**Ctrl** + **Y**	Restores an action in the open project which has been undone before.
Cut	**Ctrl** + **X**	Cuts selected parts of the **Model** view and transfers the selection to the clipboard.
Copy	**Ctrl** + **C**	Copies selected parts of the **Model** view and allows to paste these parts within the same EcoStruxure Machine Expert Twin instance.
Copy (Clipboard)	-	Copies the selection to the clipboard and allows for pasting between separate EcoStruxure Machine Expert Twin instances.
Paste	**Ctrl** + **V**	Pastes the content from the clipboard at the cursor position.

Select Section

Command	Shortcut	Description
Select All	**Ctrl** + **A**	Selects all components in the **Model** view.

View Menu

Commands of the View Menu

The **View** menu is grouped in different sections:

Windows Section

Command	Shortcut	Description
Windows	-	Select the views from the list that should be displayed in the EcoStruxure Machine Expert Twin screen. The selection is saved in the general settings. When you open EcoStruxure Machine Expert Twin next time, the selected views will be displayed.
Reset	-	Resets the views within the EcoStruxure Machine Expert Twin screen to the default state in terms of location and state as indicated in the section Default Screen, page 16.

Colors Section

Command	Shortcut	Description
Background	-	Select the color used for the background in the **Model** view.
Assembly	-	Select the color used for the components you have placed in the **Model** view. You can select different colors for: • **Assembly**: Color of the assemblies. • **Selection**: Color of selected components. • **Locked**: Color of locked components.

Scene Section

Command	Shortcut	Description
Rendering	-	Switch the rendering function **On** or **Off** by sliding the control to the right or to the left. When rendering is deactivated, no modifications are displayed in the scene.
Lock / **Unlock**	-	Lock either the entire scene or the selected assemblies to help avoid inadvertent actions or unintended modifications.
Sections	-	Opens a list of sections you created and displays the available options: • **Show/hide Section** • **Lock/unlock Section** • **Enable/disable Section**

Floor Section

Command	Shortcut	Description
AutoCad	-	Opens a **Select AutoCAD Files** dialog box that allows you to import *.dwg* or *.dxf* files.

Debug Section

Command	Shortcut	Description
Level	-	This command allows you to select the following debug levels: • **None** (default) • **Debug**: Additional information is provided in the **Logs** view. • **Detailed**: Graphical indications, applied to the objects in the scene, that allow you to visualize interactions and forces applied to those objects, allowing further investigation of the physics of the simulation.

Camera Section

Command	Shortcut	Description
Reset	**F5**	Resets the camera to the default mode.
Orthogonal	**F12** for the **Top** view	Displays the **Model** view without perspective. This allows you to visualize the scene from different angles and to align your components accurately. The following **Orthogonal** views are available: • **Top** • **Front** • **Left** • **Back** • **Right** To return to the default view with perspective, move the camera or click the **View > Reset** button.
Fit	-	Displays all components in the **Model** view simultaneously.
Camera [1...9]	**Ctrl** + **Shift** + [**1**...**9**]	Allows you to save up to 9 different camera positions by pressing **Ctrl** + **Shift** + a number between 1 and 9. To go to one of the saved positions, click the **Camera** button with the corresponding number or the shortcut **Ctrl** + the corresponding number.

Command	Shortcut	Description
Clear	-	Resets the saved camera positions.
Export	-	Allows you to export the camera positions to an XML file.
Import	-	Allows you to import a camera position from an XML file.
Follow Load	-	Follows a selected load on the way through the scene.

Shadow Section

Command	Shortcut	Description
Shadow	-	Move the slider from **Off** to **Solid** to determine how much of a shadow the production line casts.
Lighting	-	Select from the list where the light source is placed in the scene. The following places are available: • **Top** • **Front** • **Back** • **Left** ◦ **Perpendicular** ◦ **Front** ◦ **Back** • **Right** ◦ **Perpendicular** ◦ **Front** ◦ **Back**

Visibility Section

Command	Shortcut	Description
Motors	**Ctrl** + **M**	Click this toggle button to switch the visibility of motors on and off.
Label	**Ctrl** + **E**	Click this toggle button to switch the visibility of labels on and off. Labels are displayed as tooltips when hovering over an assembly in the **Model** view.

Settings Menu

Commands of the Settings Menu

Click the **Settings** button to open the view for configuring the settings for the different functions and features. It is grouped in different sections:

Camera Section

The **Camera** area allows you to configure shortcut keyboard keys for moving the cameras.

By default, the **Use Standard Configuration** check box is not selected. To define your own shortcuts, enter the key you want to use per camera movement in the different lines.

To use the following default keys, select the **Use Standard Configuration** check box:

Camera movement	Default keyboard key
Forward	**W**
Backward	**S**
Pan Left	**A**
Pan Right	**D**
Rotate Left	**Left Arrow**
Rotate Right	**Right Arrow**
Zoom In	**Up Arrow**
Zoom Out	**Down Arrow**

Snapping Section

The **Snapping** area allows you to enter the radius (in meters) in which the assembly snaps to the nearest assembly automatically while double-clicking the assembly and holding down the **Ctrl** key.

Globalization Section

The **Globalization** area allows you to select the **Measurement System**: **Metric** or **Imperial**.

Pointer Section

Check box	Setting
Keep the selection when placing a new assembly	If the check box is not selected, the new assembly continues to follow the arrow pointer when you move it around the scene. If the check box is selected, the new assembly is placed at the position that you click first.
To be able to move and rotate you have to hold down the left mouse button when this feature is enabled.	Select this check box to help avoid moving the assembly unless you hold down the left mouse button while turning it.
Invert rotation	Change the way you move across the screen with the mouse.

Copy/Paste Section

Function / Feature	Setting
Mode	Select where to insert an assembly you copied: • **Mouse**: The copied assembly is inserted at the position of the mouse. • **Offset**: The copied assembly is inserted next to the source object.
Snap	Select this check box to snap a copied assembly to the nearest assembly. As a prerequisite, the assembly must be equipped with snapping points.
Continuously Copying	Select this check box to place assemblies continuously with an offset related to the last assembly you pasted.

Statistics Section

Function / Feature	Setting
Random Seed	Click the up or down buttons to set the number you wish to use to initialize a pseudo-random sequence.
Warn on determinism lost	Select this check box to display advisory messages when the random sequence differs from the previous random sequence.

Visibility Section

Check box	Setting
See assemblies being constructed	Select this check box to see how the scene builds up when you load it instead of displaying all components at once.
Display warning signs in locked mode	Select this check box to display advisory messages when the scene is locked.

Tools Menu

Commands of the Tools Menu

The **Tools** Menu is grouped in different sections:

Capture Section

Command	Description
Screen Shot	Takes a screen capture of the **Model** view that you can save to a folder of your choice in *.png* or *.jpg* format.
Assembly Shot	Takes a screen capture of the assemblies you selected in the **Model** view. You can save it to a folder of your choice in *.png* or *.jpg* format.

Measure Section

Click the **Measure** button and then click within the **Model** view to display a measuring tape. It is a good practice to switch to top view for displaying the measuring tape.

To measure the distance between different components, hold down the **Ctrl** key to snap the tape to snap points. An alternative method, which is not as accurate, is to drag the arrows of the tape.

Model Menu

Commands of the Model Menu

The **Model** menu is grouped in different sections:

Time Section

Command	Shortcut	Description
Play	-	Starts the physical simulation.
Scale	-	Click to open the list of available execution speeds for the scene and select a speed rate. **NOTE:** Selecting an execution speed higher than 1 can lead to inaccuracies. The option **1x** sets the execution speed to real time which is indicated by **1** displayed in the status bar below the **Model** view. Alternatively, click the button **1:1** for real time setting.
Increase	-	Click to increase the execution speed by 0.1.
Decrease	-	Click to decrease the execution speed by 0.1.
Infinity	-	Click to set the execution speed to 10. **NOTE:** Selecting an execution speed higher than 1 can lead to inaccuracies.
1:1	-	Click to set the execution speed to real time.
Reset	**Ctrl** + **R**	Click to reset time and machines to 0 and to delete the loads. **NOTE:** To reset the recording of events, click the **Reset** button.
Single Step	-	Switch the **Single Step** function **On** by sliding the control to the right to pause after every time step.

Event Recorder Section

The commands provided in this area allow you to record events that occur during connection to a controller and save this recording to a separate **.events* file. This file can be replayed to allow for reviewing the behavior of the scene or to perform repetitive tests with a fixed timing without a controller being connected.

Command	Shortcut	Description
Record / **Stop** **Replay**	-	Connect to a controller and click to start / stop the recording of events being displayed in the **Model** view. Events include, for example, incoming TCP/IP messages, signals from a controller (controller outputs), manual feeding of loads. After you have loaded a **.events* file, click the **Replay** button to start replaying the recording. While replaying an events file, further information on the events is provided in the status bar at the bottom of the **Model** view.
Load	-	Click to load event recordings. As a prerequisite for replaying a recording, do the following: • Disconnect from the controller from which you recorded the events. • Click the **Reset** button from the **Time** area to reset the scene.
Save	-	Click to save event recordings in **.events* format.
Options	-	Click to open the options for event recordings: • **Pause when buffer is empty** • **Display the time for next recorded event**

Statistics Section

Command	Shortcut	Description
Snapshot	-	Click to open a **Save As** dialog box allowing you to export the information about the scene and to save them as *.csv* file.
Info	-	Opens a **Model Information** window that provides information about the number of **Assemblies**, **Motors**, **Parts**, **Connections**, **Outputs**, **Inputs**, **PhysX** available in the scene you are playing.

Selectables Section

Command	Shortcut	Description
Filter	-	Click to open the list of components of the scene that can be displayed / hidden: **Loads**, **Assemblies**, **Motors**, **Nodes**.

Loads Section

Command	Shortcut	Description
Move	-	Click to prevent loads from being moved.
Eat	-	Click to automatically delete loads that have fallen to the floor.

Routes Section

Command	Shortcut	Description
Auto / **Man**	-	Click to select whether you want EcoStruxure Machine Expert Twin to calculate routes on the assembly line if more than one option is available, or if you want to do it manually.

Kinematization Menu

Commands of the Kinematization Menu

The **Kinematization** menu is grouped in different sections:

- **Create** Section, page 31
- **Assembly** Section, page 39
- **URDF** Section, page 40
- **Smart Cams** Section, page 40

It allows you to create body assemblies and kinematic axes, as well as import CAD files and create customized kinematics with these elements, grouping them into container assemblies. By default, they are inserted at the center of the **Model** view, represented by the origin of the global coordinate system. If the value of the Z coordinate of the global coordinate system is 0, an offset of 500 mm is added for the new objects in the Z direction to help ensure that new objects are visible and not hidden or obscured by the workspace floor.

Create Section

The **Create** section allows you to create body assemblies of the selected type within the scene at the center of the **Model** view represented by the origin of the global coordinate system. When a body assembly is selected, its local coordinate system is displayed in the scene and its properties are displayed in the **Properties** view, page 32.

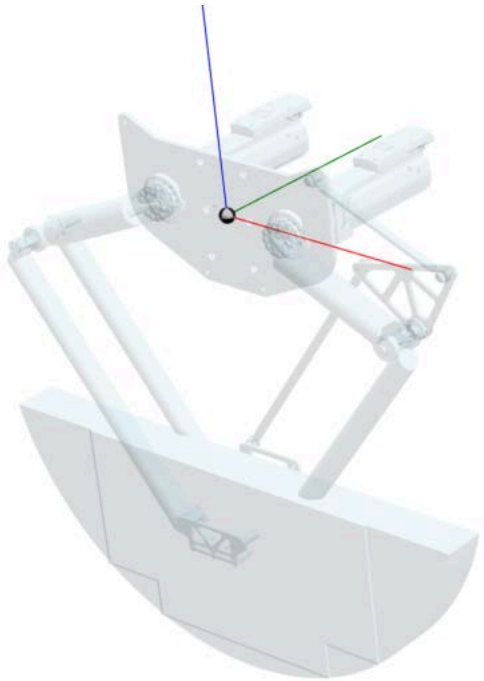
Command	Description
Box	Click to add a box to the scene at the center of the **Model** view represented by the origin of the global coordinate system.
Cylinder	Click to add a cylinder to the scene at the center of the **Model** view represented by the origin of the global coordinate system.
Sphere	Click to add a sphere to the scene at the center of the **Model** view represented by the origin of the global coordinate system.
CAD/Convex	Click to import a **.dae*, **.stl*, **.stp*, **.step*, **.sldprt* or **.CATPart* file and to add the CAD drawing together with the convex mesh at the center of the **Model** view represented by the origin of the global coordinate system. **NOTE:** In case your CAD file does not contain information on units (such as millimeter or inch), EcoStruxure Machine Expert Twin by default interprets the dimensions of objects as meter. To achieve a correct interpretation of dimensions, rescaling in your CAD tool may be required before importing the file into EcoStruxure Machine Expert Twin. For further information, refer to Importing a CAD File, page 35.
Kinematic Axis	Click to add a kinematic axis at the center of the **Model** view represented by the origin of the global coordinate system. Alternatively, select one or multiple body assemblies before clicking the **Kinematic Axis** button. One kinematic axis is then created as parent node per selected body assembly. A kinematic axis allows you to move the associated body assembly or imported CAD object within the scene. The motion can be controlled by controller signals or a motor can be added as sub device to the kinematic axis to generate motion. For further information, refer to Properties of a Kinematic Axis, page 36.

Properties of Body Assemblies

To customize a body assembly you have added to your scene using the **Kinematization** menu, select it in the **Model** view and configure your settings in the **Properties** view.

With the **Dynamics > Parameters**, configure the following:

Parameter	Description
Type	Select from the list: • **Bodiless**: The body assembly is displayed only for visualization purposes. No interaction with other objects in the scene or with the physics is performed. • **Rigid**: The body assembly interacts with other objects in the scene but is not subjected to gravitational forces. • **Physics**: The body assembly interacts with other objects in the scene and is subjected to gravitational forces.
Min Position Iterations	Enter the minimum number of iterations executed by the physical simulation to calculate the position of the body assembly based on the forces acting on it. Default value: 4
Min Velocity Iterations	Enter the minimum number of iterations executed by the physical simulation to calculate the velocity of the body assembly based on the forces acting on it. Default value: 1
Friction	Click the **Friction** arrow to open the following list of parameters that define the friction properties for the body assembly. • **Dynamic**: Dynamic friction is the frictional force encountered when two objects are in motion relative to each other. • **Static**: Static friction is the frictional force that needs to be overcome when an object starts moving, for example, after a reset. • **Coefficient**: This is a selection for the type of friction you want to apply. Select a value from the list: ◦ **None**: No friction ◦ **Slippy**: Low friction ◦ **Smooth**: Medium friction ◦ **Sticky**: High friction ◦ **Custom**: Allows you to define customized friction parameters. **NOTE:** If **Custom** is selected, enter a value of your choice. For the predefined friction types, the value is not editable.
Center Of Mass	The center of mass is a specific point at which the mass of the body can be considered to be concentrated. By default, the center of mass is set at **0:0:0 mm**. The center of mass position is relative to the origin of the geometry of the body assembly. For primitive geometries, such as box, cylinder, and sphere the origin is located at the center of the volume. In case of convex geometries (3-D CAD models), the origin is specified in the file. To shift the center of mass, edit the X, Y, and Z coordinate of the local coordinate system.

Parameter	Description
Center Of Mass Visualization	Select this option to display the center of mass of the selected body assembly in the scene when the default view is selected. It is represented by a sphere containing the center of mass symbol that is displayed relative to the origin of the body assembly. 
Weight	Enter the weight of your body assembly (in kg). Default value: 1 kg
Collision	The property is available if the parameter **Type** is set to **Rigid** or **Physics**. It allows you to enable or disable collision detection with body assemblies in the scene without impact on the collider. It is mainly used for **Physics Joints**.
Gravity	The property is available if the parameter **Type** is set to **Physics**. It allows you to enable or disable the effect of gravity on the selected body assembly. It is mainly used for **Physics Joints**.

With the **Geometry > Parameters**, configure the following:

Parameter	Description
Type	Select the type of your body assembly and configure the corresponding sub parameters. You can change the type whenever required. • **Box** ◦ **Length** ◦ **Height** ◦ **Width** • **Cylinder** ◦ **Length** ◦ **Radius** • **Sphere** ◦ **Radius** • **CAD/Convex** ◦ **Scale**: Enter the scale factor to resize a CAD drawing with respect to the original size. The default value is 1 (no scaling).
Local Position	Edit the X, Y, and / or Z coordinate to modify the position of the body assembly with reference to the origin of its own assembly coordinate system. Default value: 0:0:0 mm

Parameter	Description
Local RotX	Enter a rotation angle in ° to rotate the body assembly around the X axis of its assembly coordinate system. Default value: 0°
Local RotY	Enter a rotation angle in ° to rotate the body assembly around the Y axis of its assembly coordinate system. Default value: 0°
Local RotZ	Enter a rotation angle in ° to rotate the body assembly around the Z axis of its assembly coordinate system. Default value: 0°
Visible	Clear the check box to hide the body assembly from the scene. With the check box cleared, the body assembly is not displayed. When it is selected (for example, in the **Properties** view) its assembly coordinate system is displayed along with the body assembly.

With the **Global Pose** parameters, configure the following:

Parameter	Description
Position	Edit the X, Y, and / or Z coordinate to modify the position of the body assembly within the scene. Default value: 0:0:0 mm
RotX	Enter a rotation angle in ° to rotate the assembly and its local assembly coordinate system around the X axis. Default value: 0°
RotY	Enter a rotation angle in ° to rotate the assembly and its local assembly coordinate system around the Y axis. Default value: 0°
RotZ	Enter a rotation angle in ° to rotate the assembly and its local assembly coordinate system around the Z axis. Default value: 0°

Select the **Position > Locked** check box to lock the position of the body assembly to help prevent it from being moved within the scene.

The **Scripts > Event** parameters allow you to configure programming code that is executed when a selectable event is detected on the body assembly:

Parameter	Description
Events	Select one of the following **Events**: • **Deselect** • **DoubleClick** • **Reset** • **Select** Click the **Source > ...** button of the event to open the **Script** view, page 82.

The **Visualization** parameters allow you to configure programming code that is executed when a selectable event is detected on the body assembly:

Parameter	Description
Visible	Clear this check box to hide the body assembly in the scene.
Color	If the **Visible** check box is selected, select a color for the body assembly.

Importing a CAD File

EcoStruxure Machine Expert Twin supports the import of files with the file extensions **.dae* (Collada file), **.stl*, **.stp*, **.step*, **.sldprt* (SolidWorks) or **.CATPart* (Catia).

NOTE: In case your CAD file does not contain information on units (such as millimeter or inch), EcoStruxure Machine Expert Twin by default interprets the dimensions of objects as meter. To achieve a correct interpretation of dimensions, rescaling in your CAD tool may be required before importing the file into EcoStruxure Machine Expert Twin.

To add a CAD drawing together with the convex mesh at the center of the **Model** view represented by the origin of the global coordinate system, proceed as follows:

Step	Action
1	Click the **CAD/Convex > Import** button from the **Kinematization** menu. **Result**: The **Import CAD** dialog box is displayed.
2	Browse to the CAD file and click **Open**. **Results**: • If you have selected an **.stl* file for import, the content of the file is displayed as new assembly in the scene and one new node is added to the **Assemblies** tree of the **Solution Explorer**. • If you have selected a **.dae*, **.stp*, **.step*, **.sldprt* or **.CATPart* file for import, a message is displayed requesting you to decide whether you want to import this file as one assembly to the scene (proceed with step 3a) or to split it into its sub-assemblies (proceed with step 3b).
3a	To import the CAD file as one assembly to the scene, click **No**. **Result**: The content of the file is displayed as new assembly in the scene and one new node is added to the **Assemblies** tree of the **Solution Explorer**.
3b	To split the CAD file into its sub elements and to import them as individual assemblies, click **Yes**. **Result**: The content of the file is displayed as new assembly in the scene and several new nodes are added to the **Assemblies** tree of the **Solution Explorer** representing the sub elements that exist in the CAD file.

To display the convex mesh in the scene, run the command **Level > Detailed** from the **Debug** section of the **View** menu, page 25.

Updating a CAD File

EcoStruxure Machine Expert Twin allows you to update CAD files you had split on import into sub elements that are represented as individual assemblies.

To update a CAD file, proceed as follows:

Step	Action
1	Save the EcoStruxure Machine Expert Twin before starting the update process. If you have not saved your project, you will be prompted to save before the import is started.
2	Select the previously imported CAD file in the **Solution Explorer**.
3	Click the **CAD/Convex > Update** button from the **Kinematization** menu. **Result**: The **Update CAD** dialog box is displayed.
4	Browse to the updated CAD file, select it and click **Open**. **Result**: The **Update CAD** dialog box is displayed and lists the differences between the two file versions. On the left-hand side, it lists the sub elements as they are available in EcoStruxure Machine Expert Twin and displayed in the **Solution Explorer** view. On the right-hand side, the sub elements are listed that are available in the CAD file on the file system.
5	Compare the differences and decide which modifications you want to accept or ignore. To ignore a modification of an element, click the minus button at the right end of the respective line and select the **Ignore** command. **Result**: The selected line or group of elements is displayed greyed and will not be updated.
6	Click the **Update** button to start updating the CAD file.

CAD file comparison is performed locally using information stored within the model without relying on external hosting services. As a result, updating a CAD file overwrites the present model information. The CAD update functionality is not available for files with subnodes that share the same name under a common parent node. In this case it is not possible to distinguish and compare versions.

Properties of a Kinematic Axis

To customize a kinematic axis you have added to your scene using the **Kinematization** menu, select it in the **Model** view and configure your individual settings in the **Properties** view.

The **Coordinate Axes > Properties** parameters allow you to configure the local coordinate system of the kinematic axis:

Parameter	Description
Scale	Allows you to scale the axes of the assembly coordinate system of the kinematic axis that is selected. Default value: 1.5
Visible	Clear this check box to hide the local coordinate system of the selected kinematic axis in the scene.

The **Global Pose** parameters allow you to configure the kinematic axis with reference to the global coordinate system of the scene:

Parameter	Description
Position	Edit the X, Y, and / or Z coordinate to modify the position of the kinematic axis. Default value: 0:0:0 mm
RotX	Enter a rotation angle in ° to rotate the kinematic axis around the X axis of the global coordinate system. Default value: 0°
RotY	Enter a rotation angle in ° to rotate the kinematic axis around the Y axis of the global coordinate system. Default value: 0°
RotZ	Enter a rotation angle in ° to rotate the kinematic axis around the Z axis of the global coordinate system. Default value: 0°

From the **Motion > Type** list, select the following options:

Parameter	Description
Fixed	The kinematic axis is fixed and the associated body assemblies are not moved. This is the default setting.
Translation	The kinematic axis exerts a translational movement on the associated body assembly.
Rotation	The kinematic axis exerts a rotational movement on the associated body assembly.

With **Motion > Type** selected, the following parameters are available:

Parameter	Description
Axis	Select the direction of movement based on the assembly coordinate system: • **X** • **Y** • **Z** **NOTE:** The assembly coordinate system can be rotated which can have the effect that the directions differ from the global coordinate system.
Invert Direction	Select the **Invert Direction** check box to move the kinematic axis by default in the negative direction.
Drive Type	The options add a **Motor** as sub node to the **Kinematic Axis** node. Select an option from the list to define the controller input that is valid for the motor: • **Position**: A position value for the motor (in **Units**) is provided by a variable from the controller. • **Velocity**: A velocity value for the motor (in **Units**) is provided by a variable from the controller. • **Forward/Backward**: Boolean forward/backward signals are provided by a variable from the controller. • **Custom**: Select this option to use customized motors or positioners. If they meet the requirements defined in Customized Motors or Positioners, page 39, they are available for selection in the **Custom Drive** list and can be added to the selected kinematic axis. The motor is configured with the parameter **Drive** in this list of **Motion** parameters.
Custom Drive	Click the arrow to open a list of customized motors or positioners that meet the requirements defined in Customized Motors or Positioners, page 39. Select an entry to add it to the selected kinematic axis.
Units	Select the units of movement: • For translational movement: ◦ **Millimeters** ◦ **Meters** • For rotational movement: ◦ **Degrees** ◦ **Radians**
Limits	To limit the movement, activate the option **Use Limits** and configure the parameters (the units configured with the **Units** parameter are used): • **Min. Limit** • **Max. Limit** • **PLC Input - Min. Limit** • **PLC Input - Max. Limit**
Drive	Configure the motor according to the selection for the parameter **Drive Type**. The motor parameters are configured as described in the *How to Use Device Catalogs User Guide*. Alternatively, you can right-click the motor node in the **Solution Explorer** and control the motor manually with the **Stop**, **Start**, **Forward**, **Backward** commands

Select the **Position > Locked** check box to lock the position of the kinematic axis to help prevent it from being moved within the scene.

The **Scripts > Event** parameters allow you to configure programming code that is executed when a selectable event is detected on the kinematic axis:

Parameter	Description
Events	Select one of the following **Events**: • **Deselect** • **DoubleClick** • **Reset** • **Select** Click the **Source > ...** button of the event to open the **Script** view, page 82.

The **Visualization** parameters allow you to configure programming code that is executed when a selectable event is detected on the kinematic axis:

Parameter	Description
Visible	Clear the check box to hide the kinematic axis in the scene.
Color	If the **Visible** check box is selected, select a color for the kinematic axis.

Customized Motors or Positioners

You can configure customized motors (to move the kinematic axis by speed) or positioners (to move the kinematic axis by controller inputs). To make them available in EcoStruxure Machine Expert Twin, select the **Drive Type > Custom** option. Your motors and positioners catalogs or plugins will be displayed for selection in the **Custom Drive Type** list if they fulfill the following prerequisites:

- Motors use inheritance from *Experior.Core.Motors.Electric*.
- Positioners use inheritance from *Experior.Plugin.Kinematization.Actuators.Positioner.Positioner*.
- The catalog or plugin files are stored in the EcoStruxure Machine Expert Twin installation folder.

Assembly Section

Command	Description
Assembly Container	Click the **Assembly Container** button to create an empty assembly container. It allows you to group body assemblies and to create parent-child relationships. A new node **Container** is added to the **Solution Explorer** view. When this node is selected, the empty assembly is represented by a local coordinate system in the scene.
Tool Container	Click the **Tool Container** button to create an empty tool container. It allows you to group body assemblies and to create parent-child relationships. Note that this container type is exclusively to be attached to robotic assemblies using the **Tool Manager > Attach New Tool > Custom Tool** command. A new node **Custom Tool** is added to the **Solution Explorer** view. When this node is selected, the empty assembly is represented by a local coordinate system in the scene.

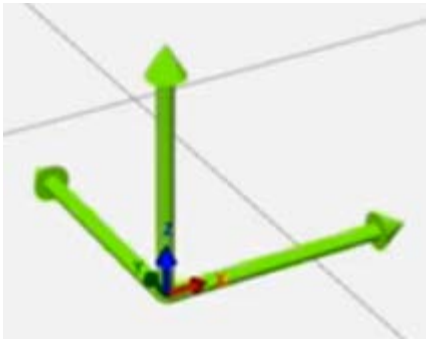
Command	Description
Attach	Creates parent-child relationships between assemblies or kinematic axes. To achieve this, select a container or a body assembly or a kinematic axis in the scene or in the **Solution Explorer**, hold down the **Ctrl** key and select another body assembly or a kinematic axis. Click the **Attach** button. As a result, the second assembly is displayed as a sub node of the first assembly in the **Solution Explorer** view. In the scene, the two assemblies act as one assembly. The relative relationship is fixed and is maintained while moving them within the scene. You can attach more body assemblies to the same parent assembly or you can create another hierarchy level by selecting the child assembly in the scene or in the **Solution Explorer**, holding down the **Ctrl** key and selecting another body assembly.
Unattach	Removes a parent-child relationship between assemblies and kinematic axes. To achieve this, select an assembly in the scene or in the **Solution Explorer** that is a child of another assembly. Click the **Unattach** button to untie this relationship. As a result, the former child assembly is now displayed as a node in the root level of the **Solution Explorer** view. In the scene, there is no longer a fixed relative relationship between the two assemblies.

URDF Section

Command	Description
Import	Click to import a **URDF** (Unified Robotics Description Format) file or alternatively an XML (eXtensible Markup Language) file that contains the adequate information. Browse to the file in the **Importing URDF File** dialog box and click **Open**. **Result**: The imported robot is displayed in the scene and a new node is created in the **Solution Explorer** view with sub nodes representing the body assemblies and kinematic axes the robot consists of.

Smart Cams Section

The smart cams function allows you to import .asc files created with the Motion Sizer that describe cubic spline interpolated profiles. For a detailed description of this file type, refer to the *Interpolated Profile File Format (.asc)* chapter of the Motion Sizer Online Help. A position profile is imported as a subnode of a selected assembly and modifies the pose (the position and orientation) of the assembly according to a master axis that is linked to this cam. One or more master axes can be configured in a project and are displayed as assembly nodes in the **Solution Explorer** view. Each cam must be linked to one master axis. The master axis acts as a virtual shaft and defines the reference position for the subordinate cams.

Command	Description
Master Axis	Click to create a **Master Axis** in the **Model** view at cursor position. The master axis acts as a virtual shaft that defines the reference position for subordinate cam axes. The master axis is represented in the **Model** view by a local coordinate system that is by default displayed in green (also refer to the global EcoStruxure Machine Expert Twin coordinate system and the local assembly coordinate systems described in chapter Information About the Coordinate System, page 17).  In the **Solution Explorer** view, a **Master Axis** node is created. For configuring a master axis, refer to Properties of a Master Axis, page 42
Create Cam	Click to create a cam. As a prerequisite for creating a cam, select an assembly of the following type: • **Cobot** (6 cams allowed) • **Delta Robots** (3 or 4 cams allowed) • **Kinematic Axis** (1 cam allowed) • **Scara** (4 cams allowed) • **Servo Belt** (1 cam allowed) For configuring a cam, refer to Properties of a Cam Diagram, page 45.
Remove Cam	Click to remove a cam selected in the **Model** view or in the **Solution Explorer** view.

Properties of a Master Axis

To customize a master axis you have added to your scene using the **Kinematization** menu, select it in the **Model** view or in the **Solution Explorer** view and configure your individual settings in the **Properties** view.

The **Cams > Items** parameters list the cam diagrams that are linked to the master axis. On the left-hand side of the list, the consecutive number of the cam diagram is displayed, starting with `[0]`. On the right-hand side, the name of the cam diagram is displayed as defined in the Properties of a Cam Diagram, page 45 plus the assembly it is assigned to (**Cobot**, **Delta Robot**, **Kinematic Axis**, **Scara** or **Servo Belt**).

Example:

```
[0]Cam1 in Kinematic Axis1
[1]Cam2 in Cobot
[2]Cam3 in Scara
[3]Cam4 in Scara
```

NOTE: Assign meaningful names to the cam diagrams to ease this linking process between master axes and cam diagrams.

The **Coordinate Axes > Properties** parameters allow you to configure the local coordinate system of the master axis:

Parameter	Description
Scale	Allows you to scale the axes of the assembly coordinate system of the master axis that is selected. Default value: 1.5
Visible	Clear this check box to hide the local coordinate system of the selected master axis in the scene.

The **Global Pose** parameters allow you to configure the master axis with reference to the global coordinate system of the scene:

Parameter	Description
Position	Edit the X, Y, and / or Z coordinate to modify the position of the master axis. Default value: 0:0:0 mm
RotX	Enter a rotation angle in ° to rotate the master axis around the X axis of the global coordinate system. Default value: 0°
RotY	Enter a rotation angle in ° to rotate the master axis around the Y axis of the global coordinate system. Default value: 0°
RotZ	Enter a rotation angle in ° to rotate the master axis around the Z axis of the global coordinate system. Default value: 0°

The **Motion > Control Mode** list allows you to select a control mode to modify the position over time.

Parameter	Description
Start/Stop	The master axis is operated in **Start/Stop** mode.
Jog	The master axis is operated in **Jog** mode.
Velocity	The master axis is operated in **Velocity** mode.

With **Motion > Control Mode > Start/Stop** selected, you can right-click the master axis in the **Model** view and run the commands **Forward** and **Backward** from the contextual menu. The following **Controller** parameters are available:

Parameter	Description
Speed	Enter a velocity for the movement of the master axis in deg/s.
Enable Acceleration	Select the option to enable acceleration control and configure the following parameters: • **Ramp Up**: Enter the maximum time (in ms) to elapse until the master axis has reached the configured **Speed** after the **Start** command. • **Ramp Down**: Enter the maximum time (in ms) to elapse until the master axis has reached standstill after the **Stop** command.
PLC Output - Invert Direction	Configure an output variable from the controller to invert the motion direction of the master axis. When TRUE, the default motion direction is inverted and motion is executed in the negative direction. • **Size**: Indicates the data type of the variable (BOOL). • **Description**: Enter a describing text. • **Symbol**: Select the output variable of the controller. The list is provided when a connection to the controller is established.
PLC Output - Start/ Stop	Configure an output variable from the controller for starting and stopping the master axis. When TRUE, the master axis is started. • **Size**: Indicates the data type of the variable (BOOL). • **Description**: Enter a describing text. • **Symbol**: Select the output variable of the controller. The list is provided when a connection to the controller is established.

With **Motion > Control Mode > Jog** selected, you can right-click the master axis in the **Model** view and run the commands **Jog Forward** and **Jog Backward** from the contextual menu. The following **Controller** parameters are available:

Parameter	Description
Step	Enter a step width for the movement of the master axis in degrees.
PLC Output - Invert Direction	Configure an output variable from the controller to invert the motion direction of the master axis. When TRUE, the default motion direction is inverted and motion is executed in the negative direction. • **Size**: Indicates the data type of the variable (BOOL). • **Description**: Enter a describing text. • **Symbol**: Select the output variable of the controller. The list is provided when a connection to the controller is established.
PLC Output - Jog	Configure an output variable from the controller for jogging the master axis. When TRUE, the master axis is started in jog mode. • **Size**: Indicates the data type of the variable (BOOL). • **Description**: Enter a describing text. • **Symbol**: Select the output variable of the controller. The list is provided when a connection to the controller is established.

With **Motion > Control Mode > Velocity** selected, you can right-click the master axis in the **Model** view and run the commands **Forward** and **Backward** from the contextual menu. The following **Controller** parameters are available:

Parameter	Description
Enable Acceleration	Select the option to enable acceleration control and configure the following parameters: • **Ramp Up**: Enter the maximum time (in ms) to elapse until the master axis has reached the configured **Speed**. • **Ramp Down**: Enter the maximum time (in ms) to elapse until the master axis has reached standstill.
PLC Output - Invert Direction	Configure an output variable from the controller to invert the motion direction of the master axis. When TRUE, the default motion direction is inverted and motion is executed in the negative direction. • **Size**: Indicates the data type of the variable (BOOL). • **Description**: Enter a describing text. • **Symbol**: Select the output variable of the controller. The list is provided when a connection to the controller is established.
PLC Output - Velocity	Configure an output variable from the controller for the velocity (in deg/s) of the master axis. The default value is 10 deg/s. • **Size**: Indicates the data type of the variable (LREAL). • **Description**: Enter a describing text. • **Symbol**: Select the output variable of the controller. The list is provided when a connection to the controller is established.

Select the **Position > Locked** check box to lock the position of the master axis to help prevent it from being moved within the scene.

The **Visualization** parameters allow you to configure programming code that is executed when a selectable event is detected on the kinematic axis:

Parameter	Description
Visible	Clear the check box to hide the kinematic axis in the scene.
Color	If the **Visible** check box is selected, select a color for the master axis.

Properties of a Cam Diagram

To customize a cam diagram you have added to your scene using the **Kinematization** menu, select it in the **Model** view or in the **Solution Explorer** view and configure your individual settings in the **Properties** view.

The **Parent** parameters allow you to select a drive type that is allowed by the parent assembly of the cam diagram:

Parameter	Description
Name	Indicates the assembly that was selected when this cam diagram was created with the **Create Cam** command. The field cannot be edited. Example: `Kinematic Axis1`
Linked to	Select the drive type that is allowed by the parent assembly from the list, for example, **Position** for kinematic axes.

The **Master Axis** parameters allow you to link the cam diagram to a master axis:

Parameter	Description
Name	Click the arrow to display a list of master axes defined in your project. Select the master axis to which you want to link your cam diagram. The motion of the cam is then in accordance with the positions received from the selected master axis.
Encoder Offset	Enter an encoder offset of the cam diagram with reference to the master axis. Range: -360...360 °

The **File** parameter allows you to import a .asc file that you have created beforehand with the Motion Sizer. For a detailed description of this file type, refer to the *Interpolated Profile File Format (.asc)* chapter of the Motion Sizer Online Help.

Parameter	Description
Name	Click the **...** button to open an **Import Cam Points File** dialog box. Browse to the .asc file you want to load, select it and click **Open**.

The **Configuration** parameters allow you to configure the cam diagram parameters.

Parameter	Description
Enabled	By default, the cam diagram is enabled. To disable, clear the check box.
Modulo Mode	By default, the option **Modulo Mode** is selected. This is suitable for cam diagrams with the same start and end point because with every new curve, the initial start point is used. For cam diagrams where the start and the end point differ, clear this option to avoid jumps in the simulation because the end point of the first curve is used as the start point of the second curve and so on.
Header	Indicates the software information that is read from line 1 of the .asc file and cannot be edited. Example: `Motion Sizer`
Version	Indicates the version information that is read from line 2 of the .asc file and cannot be edited.
Y Factor	Indicates the Y factor information that is read from line 3 of the .asc file. Click the field to configure the length of motion on the y axis (in mm for translational motion, in ° for rotational motion). The value must be greater than 0.
X Factor	Indicates the X factor information that is read from line 4 of the .asc file. Default value: 360 ° that corresponds to one full rotation of the master axis. Click the field to configure the scale of the x axis that is used to determine the x coordinate of the segment end point. The value must be greater than 0.
Y Offset	Indicates the y coordinate information of the start point of the cam segment that is read from line 5 of the .asc file. Click the field to enter an offset for the start point on the y axis (in mm for translational motion, in ° for rotational motion).
X Offset	Indicates the x coordinate information of the start point of the cam segment that is read from line 6 of the .asc file. Click the field to enter an offset for the start point on the x axis (in mm for translational motion, in ° for rotational motion). The **X Offset** parameter corresponds to the **Master Axis > Encoder Offset** parameter.
Start Slope	Indicates the slope of the position profile at the first point of the profile that is read from line 7 of the .asc file and cannot be edited.
End Slope	Indicates the slope of the position profile at the last point of the profile that is read from line 8 of the .asc file and cannot be edited.
Interpolation Mode	Indicates the interpolation mode that is read from line 9 of the .asc file and cannot be edited. • **0** = natural spline • **1** = spline on base of the slope of the borders • **2** = periodic spline
Cam Points	Indicates the [0]...[360] profile values that are read from lines 11 and the following of the .asc file. The values cannot be edited.

For further information on the .asc file content, refer to the *Interpolated Profile File Format (.asc)* chapter of the Motion Sizer Online Help

Load Handling Menu

Commands of the Load Handling Menu

The **Load Handling** menu is grouped in different sections:

- **Sensors**, page 47
- **RFID**, page 49
- **TCP Tools**, page 52
- **Vision Systems**, page 54
- **Modifiers**, page 56
- **Feeders**, page 58
- **Observers**, page 60
- **Equipment Sensors**, page 62

It allows you to create different components for load handling, such as sensors, feeders, eaters, counters. By default, they are inserted at the center of the **Model** view represented by the origin of the global coordinate system. If the value of the Z coordinate of the global coordinate system is 0, an offset of 500 mm is added for the new objects in Z direction to help avoid that they appear as if they were half sunk into the floor.

Sensors Section

The **Sensor** section allows you to create infrared sensors at the center of the **Model** view represented by the origin of the global coordinate system detecting the presence of load objects of the selected shape within the scene. When a sensor is selected, its local coordinate system is displayed in the scene and its properties are displayed in the **Properties** view, page 48.

Command	Description
Rectangular	Click to add a box-shaped sensor to the scene at the center of the **Model** view represented by the origin of the global coordinate system. When one or multiple assemblies are selected, one rectangular sensor is created per assembly with the respective dimension and at the respective position.
Cylindrical	Click to add a cylinder-shaped sensor to the scene at the center of the **Model** view represented by the origin of the global coordinate system. When one or multiple assemblies are selected, one cylindrical sensor is created per assembly with the respective dimension and at the respective position.
Spherical	Click to add a sphere-shaped sensor to the scene at the center of the **Model** view represented by the origin of the global coordinate system. When one or multiple assemblies are selected, one spherical sensor is created per assembly with the respective dimension and at the respective position.

Properties of Sensors

To customize a sensor you have added to your scene using the **Load Handling** menu, select it in the **Model** view and configure your settings in the **Properties** view.

With the **Geometry** parameters, you can configure or modify the shape of the sensor:

Parameter	Description
Type	Select the shape of your sensor and configure the corresponding sub parameters. You can change the shape whenever required. • **Rectangular** ◦ **Length** ◦ **Height** ◦ **Width** • **Cylindrical** ◦ **Length** ◦ **Radius** • **Spherical** ◦ **Radius**

With the **Filter** parameter, configure whether the sensor detects the presence of load objects:

Parameter	Description
Type	Select from the list: • **Match** (default setting): The sensor detects loads having an identifier (usually a barcode) that exactly matches the string entered in the **ID** text field below. • **Contains**: The sensor detects loads having an identifier (usually a barcode) that contains the string entered in the **ID** text field below (and can contain more characters). If the **ID** text field is empty, any kind of load object is detected.
ID	By default, this field is empty and the presence of any kind of load object is detected. To restrict detection to specific loads, enter a string that must be available in the **Identification** parameter of a load for detecting it. Also refer to Information Provided in the **Loads** View, page 85.

With the **PLC Input** parameter, configure an input variable to the controller that is set as soon as a load is detected by the sensor.

Parameter	Description
Blocked	• **Size**: Indicates the data type of the variable (BOOL). • **Description**: Enter a describing text. • **Symbol**: Select the input variable of the controller. The list is provided when a connection to the controller is established.

RFID Section

The **RFID** section allows you to create Radio Frequency Identification Devices (RFID) of a shape to be configured with the **Geometry** parameters, page 48 within the scene at the center of the **Model** view represented by the origin of the global coordinate system. When an RFID is selected, its local coordinate system is displayed in the scene and its properties are displayed in the **Properties** view.

Command	Description
RFID Reader	Click to add an RFID reader (for example, a barcode scanner) to the scene at the center of the **Model** view represented by the origin of the global coordinate system. When one or multiple assemblies are selected, one RFID reader is created per assembly with the respective dimension and at the respective position.
RFID Writer	Click to add an RFID writer (for example, a barcode writer) to the scene at the center of the **Model** view represented by the origin of the global coordinate system. When one or multiple assemblies are selected, one RFID writer is created per assembly with the respective dimension and at the respective position.

Properties of RFI Devices

To customize an RFI device you have added to your scene using the **Load Handling** menu, select it in the **Model** view and configure your settings in the **Properties** view.

With the **Filter** parameters, configure whether the RFI device detects the presence of load objects:

Parameter	Description
Type	Select from the list: • **Match** (default setting): The RFI device detects loads having an identifier (usually a barcode) that exactly matches the string entered in the **ID** text field below. • **Contains**: The RFI device detects loads having an identifier (usually a barcode) that contains the string entered in the **ID** text field below (and can contain more characters). If the **ID** text field is empty, any kind of load object is detected.
ID	By default, this field is empty and any kind of load object is detected. To restrict detection of specific loads, enter a string that must be available in the **Identification** parameter of a load for detecting it. Also refer to Information Provided in the **Loads** View, page 85.

With the **PLC Input** parameters, configure input variables to the controller that are set as soon as a load or a barcode is detected by the RFI device.

Parameter	Description
Blocked	The following parameters indicate the presence of a load object to the controller: • **Size**: Indicates the data type of the variable (BOOL). • **Description**: Enter a describing text. • **Symbol**: Select the input variable of the controller. The list is provided when a connection to the controller is established.
Barcode	The following parameters provide the barcode (identification) of a load read by the RFI device to the controller: • **Size**: Indicates the data type of the variable (STRING). • **Description**: Enter a describing text. • **Symbol**: Select the input variable of the controller. The list is provided when a connection to the controller is established.

With the **Behavior** parameters, configure the relevant number of characters of the barcode and enable and configure the writing mode.

Parameter	Description
Barcode Length	By default, the number of barcode characters read by the RFI device is 12. You can reduce the number of barcode characters transmitted by the **PLC Input > Barcode** parameter to the controller if not the entire length but only a reduced number of characters is relevant.
Enable Writing Mode	Select the option for a barcode reader to extend it by writer functions. As a result, the parameters **Write Mode** and **Automatic Barcode** are displayed below.
Write Mode	Exclusive to barcode writers: Select one of the following options: • **Automatic**: With this option selected, a new text field **Automatic Barcode** is displayed. Enter a starting value in the **Automatic Barcode** field that is written to the first load detected. For each load that is following, the value is increased by one and assigned to the loads consecutively. • **Fixed**: With this option selected, a new text field **Fixed Barcode** is displayed. Enter a fixed value in the **Fixed Barcode** field. This value without modification is written to every load that is detected. • **Controller**: With this option selected, a new parameter **Barcode** is displayed in the group of **PLC Output** parameters below.

With the **PLC Output** parameters, configure output variables from the controller for the barcode writer.

Parameter	Description
Write Trigger	Exclusive to barcode writers: When TRUE, the barcode is written to the load. The following parameters trigger the barcode writing: • **Size**: Indicates the data type of the variable (BOOL). • **Description**: Enter a describing text. • **Symbol**: Select the output variable of the controller. The list is provided when a connection to the controller is established.
Barcode	Exclusive to barcode writers: The following parameters generate a barcode (identification) that is written to the load: • **Size**: Indicates the data type of the variable (STRING). • **Description**: Enter a describing text. • **Symbol**: Select the output variable of the controller. The list is provided when a connection to the controller is established.

TCP Tools Section

The **TCP Tools** section provides commands for creating different kinds of grippers as well as trajectory plotters of a shape to be configured with the **Geometry** parameters, page 48 within the scene at the center of the **Model** view represented by the origin of the global coordinate system. When a gripper is selected, its local coordinate system is displayed in the scene and its properties are displayed in the **Properties** view. To attach a gripper to a robot, right-click the robot and run the command **Tool Manager > Attach New Tool** as described in the chapter Managing Tools Within the **Model** View, page 74.

Command	Description
Gripper	Click to add a gripper to the scene at the center of the **Model** view represented by the origin of the global coordinate system. When one or multiple assemblies are selected, one gripper is created per assembly with the respective dimension and at the respective position.
Delta Standard Gripper	Click to add a default gripper provided for delta robots to the scene at the center of the **Model** view represented by the origin of the global coordinate system.
Cobot Standard Gripper	Click to add a default gripper provided for the Lexium Cobot to the scene at the center of the **Model** view represented by the origin of the global coordinate system.
Scara Standard Gripper	Click to add a default gripper provided for the Lexium SCARA Robot to the scene at the center of the **Model** view represented by the origin of the global coordinate system.
Gripper Suppressor	Click to add a gripper suppressor for detaching a load from a gripper to the scene at the center of the **Model** view represented by the origin of the global coordinate system. When one or multiple assemblies are selected, one gripper suppressor is created per assembly with the respective dimension and at the respective position.
Trajectory Plotter	Select an assembly and click to add a trajectory plotter that displays trajectory lines with the movement of the assembly. To configure the properties of the trajectory plotter, refer to Properties of Trajectory Plotters, page 54.

Properties of Grippers and Gripper Suppressors

To customize a gripper you have added to your scene using the **Load Handling** menu, select it in the **Model** view and configure your settings in the **Properties** view.

With the **PLC Input** parameters, configure input variables to the controller that are set as soon as the presence of a load object is detected and a load has been gripped by the gripper.

Parameter	Description
Blocked	The following parameters indicate the presence of a load object to the controller: • **Size**: Indicates the data type of the variable (BOOL). • **Description**: Enter a describing text. • **Symbol**: Select the input variable of the controller. The list is provided when a connection to the controller is established.
Disable	This controller signal is exclusive to gripper suppressors. When TRUE, the gripper suppressor is deactivated. The following parameters deactivate the gripper suppressor: • **Size**: Indicates the data type of the variable (BOOL). • **Description**: Enter a describing text. • **Symbol**: Select the output variable of the controller. The list is provided when a connection to the controller is established.
Load gripped	When TRUE, a load is attached to the gripper. The following parameters indicate to the controller that a load has been gripped and is attached to the gripper: • **Size**: Indicates the data type of the variable (BOOL). • **Description**: Enter a describing text. • **Symbol**: Select the input variable of the controller. The list is provided when a connection to the controller is established.

With the **Behavior** parameters, configure a new position and orientation for the load with reference to the center of the load when the gripper is activated.

Parameter	Description
Use Attach Offset	Select this option to configure an offset with reference to the center of the load.
Position	Enter the x, y, z coordinates (in mm) where the load is to be located when the gripper is activated.
RotX	Enter an angle (in °) of rotation of the load around the X coordinate.
RotY	Enter an angle (in °) of rotation of the load around the Y coordinate.
RotZ	Enter an angle (in °) of rotation of the load around the Z coordinate.
Visible	Select this check box to display a local coordinate system at the position in the scene that is defined with the offset parameters.

With the **PLC Output** parameter, configure the output variable from the controller to activate the vacuum gripper.

Parameter	Description
Activate Vacuum	When TRUE, the vacuum gripper is activated. The following parameters activate the vacuum gripper: • **Size**: Indicates the data type of the variable (BOOL). • **Description**: Enter a describing text. • **Symbol**: Select the output variable of the controller. The list is provided when a connection to the controller is established.

Properties of Trajectory Plotters

To customize a trajectory plotter you have added to an assembly using the **Load Handling** menu, select the subnode **Trajectory Plotter** of the assembly in the **Solution Explorer** view and configure your settings in the **Properties** view.

As a prerequisite for displaying trajectory lines, start the physical simulation.

Parameter	Description
Visible	Select this check box to enable trajectory plotting.
Color	If the **Visible** check box is selected, select a color for the trajectory lines.
Retention Time	Enter a time (in milliseconds) each trajectory line is visible in the scene. Default value: 2000 ms

Vision System Section

The **Vision System** section allows you to create sensors retrieving load information regarding the position and orientation in space of a shape to be configured with the **Geometry** parameters, page 48 within the scene at the center of the **Model** view represented by the origin of the global coordinate system. When a vision system is selected, its local coordinate system is displayed in the scene and its properties are displayed in the **Properties** view.

Command	Description
Position	Click to add a vision system retrieving position information from the load to the scene at the center of the **Model** view represented by the origin of the global coordinate system. When one or multiple assemblies are selected, one vision system is created per assembly with the respective dimension and at the respective position.
Orientation	Click to add a vision system retrieving orientation information from the load to the scene at the center of the **Model** view represented by the origin of the global coordinate system. When one or multiple assemblies are selected, one vision system is created per assembly with the respective dimension and at the respective position.
Both	Click to add a vision system retrieving position and orientation information from the load to the scene at the center of the **Model** view represented by the origin of the global coordinate system. When one or multiple assemblies are selected, one vision system is created per assembly with the respective dimension and at the respective position.

Properties of Vision Systems

To customize a vision system you have added to your scene using the **Load Handling** menu, select it in the **Model** view and configure your settings in the **Properties** view.

With the **PLC Input** parameters, configure input variables to the controller that are set as soon as the presence of a load object is detected and that provide the configured position and / or orientation information. The position and orientation is relative to the origin of the sensor.

Parameter	Description
Blocked	The following parameters indicate the presence of a load object to the controller: • **Size**: Indicates the data type of the variable (BOOL). • **Description**: Enter a describing text. • **Symbol**: Select the input variable of the controller. The list is provided when a connection to the controller is established.
Position-X	The following parameters provide the position information concerning the X axis of a load to the controller: • **Size**: Indicates the data type of the variable (LREAL). • **Description**: Enter a describing text. • **Symbol**: Select the input variable of the controller. The list is provided when a connection to the controller is established.
Position-Y	The following parameters provide the position information concerning the Y axis of a load to the controller: • **Size**: Indicates the data type of the variable (LREAL). • **Description**: Enter a describing text. • **Symbol**: Select the input variable of the controller. The list is provided when a connection to the controller is established.
Position-Z	The following parameters provide the position information concerning the Z axis of a load to the controller: • **Size**: Indicates the data type of the variable (LREAL). • **Description**: Enter a describing text. • **Symbol**: Select the input variable of the controller. The list is provided when a connection to the controller is established.
Rotation-X	The following parameters provide the orientation information concerning the X axis of a load to the controller: • **Size**: Indicates the data type of the variable (LREAL). • **Description**: Enter a describing text. • **Symbol**: Select the input variable of the controller. The list is provided when a connection to the controller is established.
Rotation-Y	The following parameters provide the orientation information concerning the Y axis of a load to the controller: • **Size**: Indicates the data type of the variable (LREAL). • **Description**: Enter a describing text. • **Symbol**: Select the input variable of the controller. The list is provided when a connection to the controller is established.
Rotation-Z	The following parameters provide the orientation information concerning the Z axis of a load to the controller: • **Size**: Indicates the data type of the variable (LREAL). • **Description**: Enter a describing text. • **Symbol**: Select the input variable of the controller. The list is provided when a connection to the controller is established.

With the **Behavior** parameters, configure the information that is retrieved from the load.

Parameter	Description
Spatial Description	Select one of the following options to modify the information the vision system retrieves from the load: • **Position**: With this option selected, position information is retrieved from the load. • **Orientation**: With this option selected, orientation information is retrieved from the load. • **Pose**: With this option selected, position and orientation information is retrieved from the load.

Modifiers Section

The **Modifiers** section allows you to create objects modifying the size, color, position or the orientation of the load information of a shape to be configured with the **Geometry** parameters, page 48 within the scene at the center of the **Model** view represented by the origin of the global coordinate system. When a size or color modifier is selected, its local coordinate system is displayed in the scene and its properties are displayed in the **Properties** view.

Command	Description
Size	Click to add a sensor modifying the size of detected loads to the scene at the center of the **Model** view represented by the origin of the global coordinate system. When one or multiple assemblies are selected, one vision sensor is created per assembly with the respective dimension and at the respective position.
Color	Click to add a sensor modifying the color of detected loads to the scene at the center of the **Model** view represented by the origin of the global coordinate system. When one or multiple assemblies are selected, one vision sensor is created per assembly with the respective dimension and at the respective position.
Position	Click to add a sensor modifying the position of detected loads to the scene at the center of the **Model** view represented by the origin of the global coordinate system. When one or multiple assemblies are selected, one vision sensor is created per assembly with the respective dimension and at the respective position.
Orientation	Click to add a sensor modifying the orientation of detected loads to the scene at the center of the **Model** view represented by the origin of the global coordinate system. When one or multiple assemblies are selected, one vision sensor is created per assembly with the respective dimension and at the respective position.

Properties of Modifiers

To customize a modifier you have added to your scene using the **Load Handling** menu, select it in the **Model** view and configure your settings in the **Properties** view.

With the **PLC Input** parameter, configure the input variable to the controller that is set as soon as the presence of a load object is detected.

Parameter	Description
Blocked	The following parameters indicate the presence of a load object to the controller: • **Size**: Indicates the data type of the variable (BOOL). • **Description**: Enter a describing text. • **Symbol**: Select the input variable of the controller. The list is provided when a connection to the controller is established.

With the **Load Parameters**, configure the new target dimensions, color, or position that are assigned by the modifier to the loads.

Parameter	Description
Length	Exclusive to size modifiers: Enter the new target length that is assigned to the load (in mm).
Height	Exclusive to size modifiers: Enter the new target height that is assigned to the load (in mm).
Width	Exclusive to size modifiers: Enter the new target width that is assigned to the load (in mm).
Radius	Exclusive to size modifiers: Enter the new target radius that is assigned to the load (in mm).
Color	Exclusive to color modifiers: Click to open the color palette and select a color that is assigned to the load.
Relative Position	Exclusive to position modifiers: Edit the X, Y, and / or Z coordinate to modify the relative position that is assigned to the load. Default value: 0:0:0 mm
RotX	Exclusive to orientation modifiers: Enter a rotation angle in ° around the X axis that is assigned to the load.
RotY	Exclusive to orientation modifiers: Enter a rotation angle in ° around the Y axis that is assigned to the load.
RotZ	Exclusive to orientation modifiers: Enter a rotation angle in ° around the Z axis that is assigned to the load.

With the **PLC Output** parameter, configure the output variable from the controller to deactivate the modifier.

Parameter	Description
Disable	When TRUE, the modifier is deactivated. The following parameters deactivate the modifier: • **Size**: Indicates the data type of the variable (BOOL). • **Description**: Enter a describing text. • **Symbol**: Select the output variable of the controller. The list is provided when a connection to the controller is established.

Feeders Section

The **Feeders** section allows you to create feeders that add loads to your scene. For general information on feeders, refer to *Feeders, Loads and Eaters* in the EcoStruxure Machine Expert Twin How to Use Device Catalogs User Guide.

Command	Description
Standard Feeder	Click to add a generic (herein standard) feeder assembly to the scene at the center of the **Model** view represented by the origin of the global coordinate system. The standard feeder can create different types of predefined loads.
Custom Feeder	Select a body assembly or a container assembly and click to add a customized feeder. The feeder is placed within the scene using the position and orientation of the selected prototype body assembly. When it is selected, its local coordinate system is displayed in the scene and its properties are displayed in the **Properties** view. As a prototype for the loads that are created by the customized feeder, you can use a body assembly that has been created by importing a CAD file to EcoStruxure Machine Expert Twin as explained in Importing a CAD File, page 35. For using it as feeder prototype, you can import the CAD file as one assembly or as individual assemblies depending on the intended use.

Properties of Customized Feeders

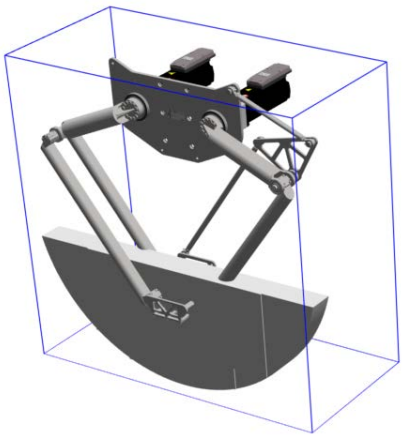
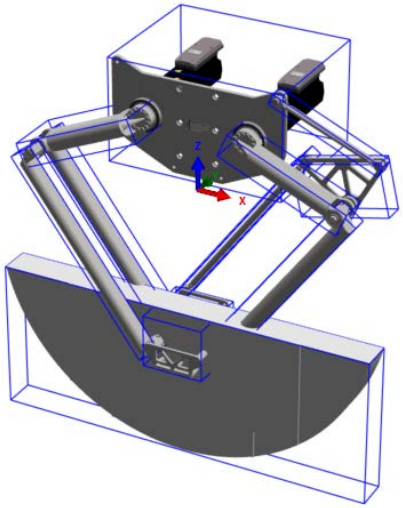
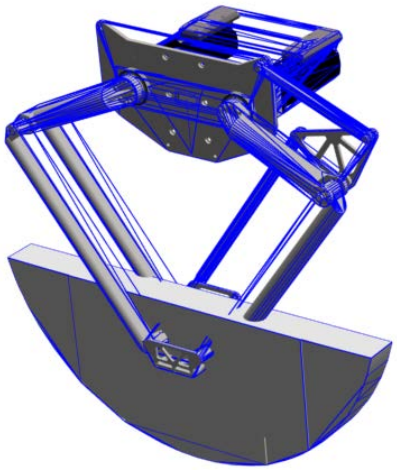
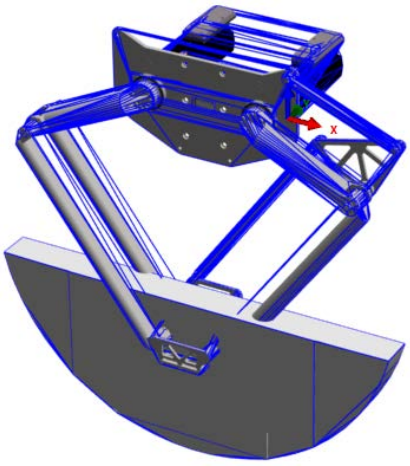
To configure a **Custom Feeder** you have added to your scene using the **Load Handling** menu, select it in the **Model** view and configure your settings in the **Properties** view.

The **Load** parameters indicate the body assembly that is used as a base for the loads to be generated and allow further configurations. This chapter only explains the properties that are specific to customized feeders. For general properties, refer to *Feeders, Loads and Eaters* in the EcoStruxure Machine Expert Twin How to Use Device Catalogs User Guide

Parameter	Description
Custom Load Assembly	Indicates the name of the body assembly that is used as a base for the loads that are generated. This base cannot be modified. To use a different body assembly as base, create another custom load feeder.
Rigid	Defines the collider geometry used by the physical simulation. • **Box** Select this option to create new loads as simple, rigid boxes. • **Convex** Select this option to create new loads with a more accurate collider geometry.

If the CAD file of the body assembly that is used as a base for the loads had been imported as several assemblies (see Importing a CAD File, page 35), each load that is generated also consists of several body assemblies. For each body assembly that constitutes the load, an individual geometry is created. The accuracy of the collider geometry of each body assembly depends on the option selected for the **Rigid** parameter.

The following figures illustrate the differences for CAD files imported as single or as multiple body assembly with **Rigid > Box** and **Rigid > Convex** selected and with the convex mesh displayed in the scene (activated by running **Level > Detailed** from the **Debug** section of the **View** menu, page 25).

Rigid option selected	CAD file imported as single body assembly	CAD file imported as multiple body assembly
Box		
Convex		

Observers Section

The **Observers** section allows you to create eaters, counters and weight scales of a shape to be configured with the **Geometry** parameters, page 48 at the center of the **Model** view represented by the origin of the global coordinate system. When an eater, counter or weight scale is selected, its local coordinate system is displayed in the scene and its properties are displayed in the **Properties** view.

Command	Description
Eater	Click to add an eater removing detected loads from the scene at the center of the **Model** view represented by the origin of the global coordinate system. When one or multiple assemblies are selected, one eater is created per assembly with the respective dimension and at the respective position.
Counter	Click to add a counter counting detected loads at the center of the **Model** view represented by the origin of the global coordinate system. When one or multiple assemblies are selected, one counter is created per assembly with the respective dimension and at the respective position.
Weight Scale	Click to add a weight scale to the scene at the center of the **Model** view represented by the origin of the global coordinate system. When one or multiple assemblies are selected, one weight scale is created per assembly with the respective dimension and at the respective position.

Properties of Observers

To customize an observer you have added to your scene using the **Load Handling** menu, select it in the **Model** view and configure your settings in the **Properties** view.

With the **PLC Input** parameter, configure the input variables to the controller that are set as soon as the presence of a load object is detected.

Parameter	Description
Blocked	The following parameters indicate the presence of a load object to the controller: • **Size**: Indicates the data type of the variable (BOOL). • **Description**: Enter a describing text. • **Symbol**: Select the input variable of the controller. The list is provided when a connection to the controller is established.
Counter	The following parameters are used for counting the number of loads that have been deleted by the eater or that have been detected by the counter: • **Size**: Indicates the data type of the variable (INT). • **Description**: Enter a describing text. • **Symbol**: Select the input variable of the controller. The list is provided when a connection to the controller is established.
Weight	Exclusive to weight scales: The following parameters provide the weight of the load (in kg) measured by the sensor to the controller: • **Size**: Indicates the data type of the variable (REAL). • **Description**: Enter a describing text. • **Symbol**: Select the input variable of the controller. The list is provided when a connection to the controller is established.

With the **PLC Output** parameters, configure the output variables from the controller for counting and deactivating the observers.

Parameter	Description
Disable	When TRUE, the observer is deactivated. The following parameters deactivate the observer: • **Size**: Indicates the data type of the variable (BOOL). • **Description**: Enter a describing text. • **Symbol**: Select the output variable of the controller. The list is provided when a connection to the controller is established.

Equipment Sensor Section

The **Equipment Sensor** section allows you to create sensors that detect the presence of static objects or equipment (such as conveyors) at the center of the **Model** view represented by the origin of the global coordinate system. When an equipment sensor is selected, its local coordinate system is displayed in the scene and its properties are displayed in the **Properties** view.

Command	Description
Rectangular	Click to add a box-shaped equipment sensor to the scene at the center of the **Model** view represented by the origin of the global coordinate system. When one or multiple assemblies are selected, one rectangular sensor is created per assembly with the respective dimension and at the respective position.
Cylindrical	Click to add a cylinder-shaped equipment sensor to the scene at the center of the **Model** view represented by the origin of the global coordinate system. When one or multiple assemblies are selected, one cylindrical sensor is created per assembly with the respective dimension and at the respective position.
Spherical	Click to add a sphere-shaped equipment sensor to the scene at the center of the **Model** view represented by the origin of the global coordinate system. When one or multiple assemblies are selected, one spherical sensor is created per assembly with the respective dimension and at the respective position.

Properties of Equipment Sensors

To customize an equipment sensor you have added to your scene using the **Load Handling** menu, select it in the **Model** view and configure your settings in the **Properties** view.

With the **Geometry** parameters, you can configure or modify the shape of the sensor:

Parameter	Description

Type	Select the shape of your sensor and configure the corresponding sub parameters. You can change the shape whenever required.
	• **Rectangular**
	  ◦ **Length**
	  ◦ **Height**
	  ◦ **Width**
	• **Cylindrical**
	  ◦ **Length**
	  ◦ **Radius**
	• **Spherical**
	  ◦ **Radius**

With the **PLC Input** parameter, configure the input variable to the controller that is set as soon as the presence of static objects is detected.

Parameter	Description
Blocked	The following parameters indicate the presence of a static object to the controller: • **Size**: Indicates the data type of the variable (BOOL). • **Description**: Enter a describing text. • **Symbol**: Select the input variable of the controller. The list is provided when a connection to the controller is established.

SmartVisu Menu

General Information

The **SmartVisu** menu allows you to generate a 3-D digital twin of the machine you configured with the DigitalTwinCommunication library in EcoStruxure Automation Expert - Motion or EcoStruxure Machine Expert. For further information, refer to the DigitalTwinCommunication Library Guide.

As a prerequisite, an OPC UA connection to a PacDrive LMC or a Modicon M262 Motion Controller must be established. For further information on creating an OPC UA connection, refer to the Communication User Guide.

Elements of the SmartVisu Menu

Element	Description
Connection ID	Select the OPC UA connection to the desired controller for which you want to create a 3-D digital twin.
Generate	Reads the settings made in the DigitalTwinCommunication library and displays a 3-D digital twin of your machine in the **Model** view. **NOTE:** The connection from EcoStruxure Machine Expert Twin to the controller is using a handshake mechanism to acknowledge that event data was received and ensure consistent information is displayed in the **Model** view (single-instance mode). In the case of multiple connections from EcoStruxure Machine Expert Twin to the same controller, the primary connection is verifying the data exchange by the handshake mechanism. Further connections are working in multi-instance mode, and can process event data simultaneously, but the handshake mechanism is not used for multiple connections. In few cases, under high usage of the communication bandwidth, full transmission of events may not be possible, and might result in events not being displayed in the instances using multi-instance mode.
Remove	Removes the assemblies and loads generated from this connection from the **Model** view.

Managing Projects with SmartVisu Assemblies

It is possible to use different SmartVisu assemblies in the same EcoStruxure Machine Expert Twin project. To add two SmartVisu assemblies to a project, proceed as follows:

Step	Action
1	Establish an OPC UA connection to a PacDrive LMC or a Modicon M262 Motion Controller. For further information on creating an OPC UA connection, refer to the Communication User Guide.
2	From the **SmartVisu > Connection ID** list, select the entry corresponding to the connection that has the **Status** = **Connected** in the **Connections** view, page 67, for example, **Connection1**.
3	Click the **SmartVisu > Generate** command. **Results**: • The 3-D digital twin of the **Connection1** machine is displayed at the center of the **Model** view. • A new subnode **SmartVisu [000 - Connection1]** is added to the **Assemblies** node of the **Solution Explorer** view, page 94.
4	Select the **SmartVisu [000 - Connection1]** node and edit the **Transformation > Position** value to move the section from the center before you insert the second SmartVisu section.
5	Establish a second OPC UA connection to a PacDrive LMC or a Modicon M262 Motion Controller.
6	From the **SmartVisu > Connection ID** list, select the entry corresponding to this second connection that also has the **Status** = **Connected** in the **Connections** view, page 67, for example, **Connection2**.
7	Click the **SmartVisu > Generate** command. **Results**: • The 3-D digital twin of the **Connection2** machine is displayed at the center of the **Model** view. • A new subnode **SmartVisu [001 - Connection2]** is added to the **Assemblies** node of the **Solution Explorer** view, page 94.
8	To save the two SmartVisu sections to a project, run the **File > Save As** command, browse to a folder, enter a file name and click **Save**.

You can open and edit projects that contain SmartVisu assemblies without the OPC UA connection being established. When you attempt to establish the connection after the SmartVisu assemblies have been modified, a **SmartVisu** dialog box is displayed indicating the changes. It allows you to reload the SmartVisu configuration from the DigitalTwinCommunication library in EcoStruxure Automation Expert - Motion or EcoStruxure Machine Expert. This overrides the modifications you performed on the project.

NOTE: Verify the modifications that are listed in the **SmartVisu** dialog box. To prevent the modifications from being overwritten, click **No** and save the project with a new name.

Auto-Generation Menu

Commands of the Auto-Generation Menu

The **Auto-Generation** menu allows you to read the configuration of a multi carrier, a robot or a servo belt and to create the corresponding assemblies after having established a connection to the controller. As a prerequisite, the IP address and the connection parameters must be configured correctly and an OPC UA connection must be established. For further information on the prerequisites, refer to:

- The *Procedure for Starting the Emulation* in the How to Emulate User Guide of the EcoStruxure Automation Expert - Motion / EcoStruxure Machine Expert online help.
- The description of *Creating Emulation Data* in the Lexium™ MC multi carrier Configuration Guide of the EcoStruxure Automation Expert - Motion / EcoStruxure Machine Expert online help.

The menu is grouped in different sections:

multi carrier Section

Command	Description
Name	Enter the name of the multi carrier object to be generated, for example, `MC_Track_1`.
Connection ID	Select the **Connection ID** from the list in accordance with the **Connections** view, page 67.
Configure Data	Click this button to see the input and output data that is exchanged cyclically between EcoStruxure Machine Expert Twin and the controller. In order to optimize controller performance, you can activate or deactivate the exchange of information as required in your application: • **Carrier TrackId** By default, the unique track ID is retrieved from the controller with the carrier information. • **Carrier Product Color** By default, the color of the product is retrieved from the controller with the carrier information. • **Carrier Collision Detection** By default, collision detection information is transmitted to the controller. • **multi carrier Error Display** By default, information on detected error states of carriers and segments is not retrieved from the controller.
Load Object	Click to establish a connection to the configured controller and to read the configuration of the multi carrier. You can also click the button to update the created assemblies after you have modified the configuration of the multi carrier.

Robots Section

Command	Description
Name	Enter the name of the robot object to be generated, for example, `T_Robot_1`.
Connection ID	Select the **Connection ID** from the list in accordance with the **Connections** view, page 67.
Load Object	Click to establish a connection to the configured controller and to read the configuration of the robot. You can also click the button to update the created assemblies after you have modified the configuration of the robot.

Servo Belts Section

Command	Description
Name	Enter the name of the servo belt object to be generated, for example, `Infeed_Belt`.
Connection ID	Select the **Connection ID** from the list in accordance with the **Connections** view, page 67.
Load Object	Click to establish a connection to the configured controller and to read the configuration of the servo belt. You can also click the button to update the created assemblies after you have modified the configuration of the servo belt.

Connections View

What's in This Chapter

Connections View

Creating Connections

The **Connections** view allows you to create connections to your controllers or higher level systems. By default, the view is empty. Right-click within the view to open the **Communication** dialog box for selecting the communication protocol for your connection.

Select a communication protocol and an entry will be displayed in the **Connections** view for this connection.

By default, the following communication protocols are supported:

Address-based communication protocols:

- **Siemens**
- **Modbus**
- **Serial (Raw)**
- **XCom**

Tag-based communication protocols:

- **Beckhoff ADS**
- **Ethernet/IP - CIP Browsable**
- **OPC UA Client**

Messages-based communication protocols:

- **STC/ETX**
- **3964R**
- **RFC 1006**

Configuring Connections

For a detailed description on how to configure an OPC UA connection, refer to the EcoStruxure Machine Expert Twin How to Use Device Catalogs User Guide.

In general, configure a connection in two steps:

1. Select the row in the list of the **Connections** view and configure the corresponding settings in the **Properties** view.
2. Right-click the row in the list of the **Connections** view to open the contextual menu providing the following commands, depending on the connection type:

Command	Description
Connect All / **Disconnect All**	Connects / disconnects all controllers simultaneously providing you have entered an IP address for each connection.
Connect / **Disconnect**	Connects to / disconnects from the selected controller.
Listen / **Disconnect**	Listens to / disconnects from the selected server. The communication protocols **RFC 1006** and **3964R** TCP/IP can act as server or client depending on the configuration in the **Properties** view.
Enable Logging	Logs the messages sent by the controller and displays them in the **Logs** view, page 83.
Truncate Log	If logging is enabled on the controller, text files (*.txt) are created in your local Windows user directory: *%localappdata%\Schneider Electric\Machine Expert Twin\1\Work \Logs* The log files are updated when the project is closed. With this option selected, the log files are cleared when the project is opened.
Import	Imports the addresses from an Excel file to the controller.
Export	Exports the addresses from the controller to an Excel file.
Alarms	Opens the **Alarm Scheme** dialog box that allows you to set alarms and which error messages they will display. For further information, refer to the description of the **Alarm Scheme** Dialog Box, page 68.
Delete	Deletes the connection.

Alarm Scheme Dialog Box

In the **Connections** section, right-click on a connection and select **Alarms** to display the **Alarms Scheme** dialog box.

Column	Description
Subscribe	Select whether or not the alarm is activated.
• **Address** • **Bit** • **Source**	These parameters define the address of the alarm.
Description	Enter the description for the alarm.

If the data exchange to the controller is address-based, add alarms when the controller is not connected. If the data exchange is tag-based, add alarms when the controller is connected. Also refer to the description of the **Alarms** view, page 87.

Catalogs View

What's in This Chapter

Catalogs View

General Information

The **Catalogs** view displays the catalogs that have been selected during start-up of EcoStruxure Machine Expert Twin in the **Select Catalog(s)** dialog box.

Catalogs are handled as DLL files that are by default provided in the EcoStruxure Machine Expert Twin installation folder. The default catalogs are extended by the prefix *Experior.Catalog*, for example *Experior.Catalog.SchneiderElectric. Accessories.dll*.

Catalogs function as a form of libraries that provide different assemblies and other items. You can also create your own catalogs using the C# programming language. To reference your own catalogs, click the browse (**...**) button in this dialog box and browse to the folder that contains your catalogs.

Selecting Objects from Catalogs

The **Catalogs** view displays the catalogs you selected in the **Select Catalog(s)** dialog box either in a **Gallery View Style** or in a **Tree View Style**. Select a catalog to see the objects it contains. To use an object in your scene, select the object, position the cursor in the **Model** view and click to place the object.

If the **Gallery View Style** is selected, objects you insert consecutively are automatically connected. For example, you can insert different types of conveyors consecutively by double-clicking the conveyor objects in the **Belts & Conveyors** catalog. As a result, each new conveyor is snapped to the end of the last conveyor extending the line of conveyors.

For further information, refer to the EcoStruxure Machine Expert Twin How to Use Device Catalogs User Guide.

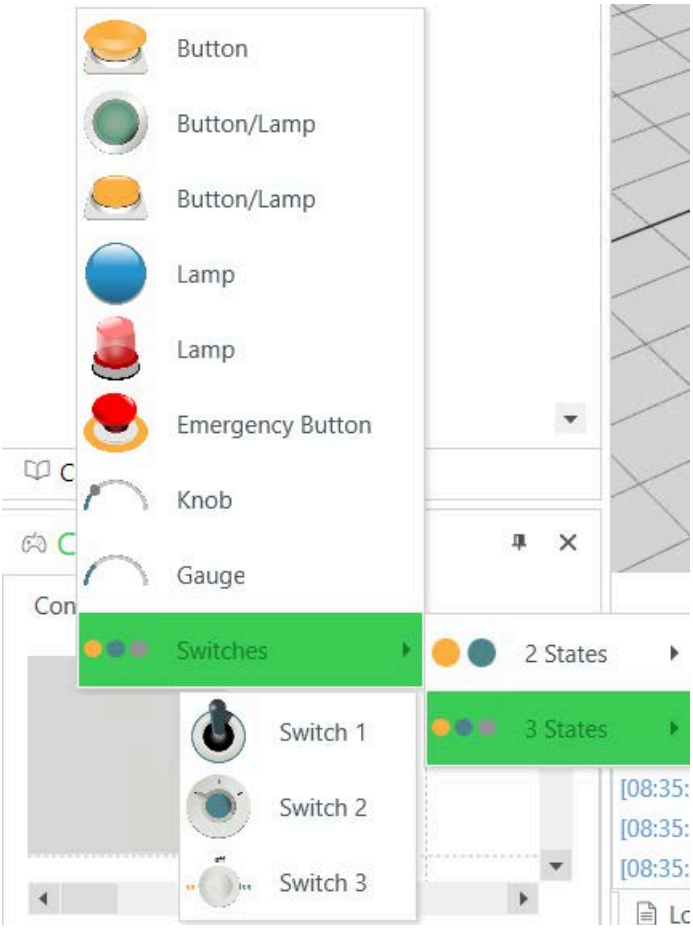
Control Panel View

What's in This Chapter

Control Panel View

Creating Controls

The **Control Panel** view allows you to insert controls for communicating with the controller. Right-click on an empty square of the **Control Panel** view to open a contextual menu that provides different types of controls:



The **Buttons** work when you press them while the **Lamps** light up when a specific controller signal is sent.

The **Knob** allows you to switch between different values which you can select in the **Properties** view.

The **Gauge** displays values sent from the controller.

The **Switches** option allows you to choose between two-state and three-state switches.

After you have selected a control, you can edit the properties in the **Properties** view such as name and color.

Copying / Pasting / Removing Controls

To copy a control, right-click it in the **Controls (n)** tab and run the command **Copy** from the contextual menu.

Right-click in the **Controls (n)** tab and run the command **Paste** to insert the control you copied.

To remove a control, right-click it in the **Controls (n)** tab and run the command **Delete** from the contextual menu.

Managing Control Panel Tabs

To manage the tabs of the **Control Panel** view, right-click the header of a **Controls (n)** tab. A contextual menu with the following commands is displayed:

Command	Description
Left / **Right** **First** / **Last**	Allows you to move a **Controls (n)** tab to the left, to the right, to the first or to the last position.
New Control Panel	Creates a new **Controls (n+1)** tab and inserts it at the right end.
Duplicate (tab name)	Copies the active tab with its content and inserts it as new tab with tab name **Copy of (tab name)** to the right of the existing tabs.
Delete (tab name)	Deletes the active tab.
Delete All	Deletes all tabs. As a result, the two empty default tabs **Controls (1)** and **Controls (2)** are provided.
Undock	Allows you to move the selected tab to another position in the EcoStruxure Machine Expert Twin screen.
Alignment	Allows you to display the tab headings not at the **Top** of the **Control Panel** view, as it is by default, but at the **Bottom**, the **Left** or the **Right** of the view.

Model View

What's in This Chapter

Model View

Overview

The **Model** view is where you display your scene and its constituent assemblies and loads. You can also graphically visualize what happens when you play the scene.

The three arrows in the middle represent the 3-D coordinate system of the **Model** view using the colors X = red, Y = green, Z = blue. For further information, refer to Information About the Coordinate System, page 17.

Zooming, Rotating, Moving the Camera View

To zoom the camera view within the **Model** view, scroll the mouse wheel or set the zoom shortcuts in the **Settings** menu, page 26.

To rotate the camera view, hold down the left mouse button, and move the mouse.

To move the camera view, hold down the right mouse button, and move the mouse.

Rotating Objects in the Scene

Select an object in the **Model** view and perform the following mouse actions for rotation around the different axes:

Mouse Action	Rotation around axis
Scroll the mouse wheel	Z
Hold down the **Ctrl** key and scroll the mouse wheel	Y
Hold down the **Alt** key and scroll the mouse wheel	X

Sections

By default, the **Model** view contains one section. The section is the container of assemblies inside a scene. The default section has a size of 50,000 mm x 50,000 mm and can be modified in the **Properties** view. The floor of the section is displayed gray by default but if you flip the scene to see it from below the floor appears transparent. In the **Properties** view, the floor can be modified regarding color and type.

Adding Sections

To add sections to the **Model** view, proceed as follows:

Step	Action
1	In the **Solution Explorer** view, right-click the **Assemblies** node and run the command **Add Section** from the contextual menu. **Results**: • In the **Solution Explorer** view, a new node **Section[n+1]** is added below the last section node. • In the **Model** view, a new section is displayed on top of the existing section. Unless you changed the dimensions or the position of the existing section you will not be able to see the new section immediately as it is directly on top of the existing.
2	In the **Solution Explorer** view, select the node of the new section, to make it active. **Result**: Further information on the section is displayed in the **Properties** view.

Modifying the Floor Using the Section Properties

You can modify what is displayed as the floor in the **Model** view by modifying the properties of the section. Proceed as follows:

Step	Action
1	In the **Solution Explorer** view, expand the **Assemblies** node and select the **Section** node.
2	In the **Properties** view, modify the properties.

- With the parameter **Section > Name** you can assign a name to the section that is displayed at the lower right corner of the floor in the **Model** view.
- To hide the selected section from the **Model** view, clear the check box **Visible**.
- Enter the **Size > Width** and **Size > Length** of your choice to modify the size of the section.
- Enter the **Transformation > Position > 0:0:0 mm** and / or the **Transformation > Yaw** (in °) to modify the position of the section in space (in a cartesian coordinate system).
- Select an option from **Assemblies** to move, lock, enable or select assemblies in the selected section.
- Select another option from **Floor > Type** to modify, for example, the color or the texture of the section displayed as floor.
- In **Grid**, you can, for example, hide the grid and enable bounds or you can import a floor design of your choice by selecting the option **AutoCAD** and importing a **.dxf* or **.dwg* file.

The properties defined here are specific to the selected section. When you open a new **Model** view, a default section with the default settings opens.

Status Bar

The status bar at the bottom of the **Model** view displays information about the scene:

Column	Description
Paused / **Running**	Indicates if the time is running (the scene is playing) or paused.
Scan	Indicates the time (in ms) it took to calculate the last step of the physical simulation. If this number becomes too high, an advisory message will be displayed on the right-hand side of this status bar.
Lock icon	Indicates whether the scene is locked.
Time displayed as [dd:hh:mm:ss]	Displays the time elapsed while playing the scene.
Time scale	Displays the time scale for playing the scene, such as: **1** = real time **2** = double real time The maximum is **10** times real time.
Information and diagnostic messages	Provides information and advisory messages that are relevant while playing the scene. The messages are color-coded according to the severity. For information on messages provided in the **Logs** view, refer to the chapter **Logs** View, page 83.

Managing Tools Within the Model View

Overview

You can attach or detach tools to or from assemblies that are available in the **Model** view by using the **Tool Manager**. To access the function, right-click the assembly in the **Model** view and run the command **Tool Manager** from the contextual menu.

The following assembly types are supported:

- Lexium Cobot
- Lexium SCARA Robot
- Delta robots
- Carriers of a Lexium™ MC multi carrier track

You can attach tools that are available in EcoStruxure Machine Expert Twin or that you create using the **Kinematization** menu, page 31.

Attaching a Tool to an Assembly

To attach a tool to an assembly, right-click the assembly in the **Model** view and run one of the following commands from the **Tool Manager** contextual menu:

- **Tool Manager > Attach Existing Tool** to attach a tool that is available in the scene but not attached to an assembly.
- **Tool Manager > Attach New Tool** to attach a new tool in two different ways:
  - Select **Tool Manager > Attach New Tool > Custom Tool** to attach a container assembly that you create with the **Tool Container** command of the **Kinematization** menu, page 39.
  - Attach a tool that is available by default, for example, by running **Tool Manager > Attach New Tool > Cobot Standard Gripper** to attach the default gripper provided for the Lexium Cobot or **Tool Manager > Attach New Tool > Delta Standard Gripper** to attach the default gripper provided for delta robots.

Attaching a New Tool to an Assembly

To attach a new tool that is provided by default by EcoStruxure Machine Expert Twin, proceed as described in this example for attaching the **Cobot Standard Gripper** to a Lexium Cobot:

Step	Action
1	Select an object in the **Cobot** catalog and add it to the **Model** view.
2	Right-click the Cobot assembly in the **Model** view and run the command **Tool Manager > Attach New Tool > Cobot Standard Gripper** from the contextual menu. **Result**: The tool is attached to the Cobot assembly in the **Model** view and is displayed in the **Solution Explorer** view as subnode **Cobot Standard Gripper** of the Cobot node.
3	Select the node **Cobot Standard Gripper** and subnodes **Vacuum Actuator**, **Vacuum Sensor** in the **Solution Explorer** view to configure the properties of the gripper in the **Properties** view according to your individual requirements.

Attaching a New Customized Tool to an Assembly

You can attach a tool container that you create with the **Tool Container** command of the **Kinematization** menu, page 39 as follows:

Step	Action
1	Right-click an assembly in the **Model** view and run the command **Tool Manager > Attach New Tool > Custom Tool** from the contextual menu. **Result**: A small black cube is displayed at the assembly in the **Model** view and in the **Solution Explorer** view, a new subnode **Custom Tool** including another subnode **Kinematization Root** is added to the node of the selected assembly.
2	Select the subnode **Kinematization Root**, open the **Kinematization** menu and attach body assemblies or container assemblies as described in **Kinematization** Menu, page 31. **Result**: The body assemblies or container assemblies are attached as a tool to the selected assembly and are displayed as subnodes of the **Kinematization Root** in the **Solution Explorer** view. This relationship as well as the **Kinematization** container assembly itself is saved with the .Experior project and is preserved when the project is closed and reopened.

Attaching an Existing Tool to an Assembly

To attach a tool that is available in the scene, proceed as follows:

Step	Action
1	Right-click an assembly in the **Model** view and run the command **Tool Manager > Attach Existing Tool** from the contextual menu. **Result**: The tools and tool containers that are available in the scene but not attached to an assembly are provided for selection.
2	Select the tool you want to attach. **Result**: A message is displayed requesting to you to decide whether you want to preserve the position and orientation of the tool with respect to the assembly. Proceed with step 3a clicking **Yes** or with step 3b by clicking **No**.
3a	Click **Yes** to preserve the position and orientation of the tool with respect to the assembly. **Result**: The tool is attached to the assembly without changing its position or orientation in the **Model** view but with an offset to the assembly. This offset is preserved and the tool is moved in accordance with the assembly changing position and or orientation within the scene.
3b	Click **No** to attach the tool to the assembly. **Result**: The tool is moved within the scene and attached to the assembly. Use the **Properties > Offsets Relative to Flange** parameters to adjust the orientation of the tool relative to the assembly.

Detaching a Tool from an Assembly

To detach a tool from an assembly, proceed analog to the example for attaching a cubic gripper to a Lexium Cobot:

Right-click the Cobot assembly in the **Model** view and run the command **Tool Manager > Detach Active Tool(Experior.Catalog.Schneider.Electric.Cobots.CubicGripper)** from the contextual menu.

This results in two separate stand-alone assemblies indicated in the **Model** view and in the **Solution Explorer** view:

- The Cobot assembly
- The gripper assembly

When you detach a **Kinematization** container assembly from an assembly, the **Kinematization** container assembly is preserved as stand-alone in the scene and is displayed in the **Solution Explorer** view as a separate **Kinematization Root[n]** node.

Preparing a Kinematization Chain for Use as Kinematization Tool

To create a kinematization chain with the **Kinematization** menu and to prepare it for use as kinematization tool, proceed as follows:

Step	Action
1	Create a kinematization chain by creating body assemblies and / or kinematic axes using the **Kinematization** menu, selecting them in the scene or in the **Solution Explorer** by holding down the **Ctrl** key and clicking the **Attach** button from the **Kinematization** menu. **Result**: A node for this kinematization chain is created in the **Solution Explorer** view.
2	Click the **Create** button in the **Kinematization** menu to create a container assembly as described in the **Assembly** Section of the **Kinematization** Menu, page 39. **Result**: An **Assembly** node is created in the **Solution Explorer** view for this empty container assembly.
3	Select the **Assembly** node in the **Solution Explorer**, hold down the **Ctrl** key, select the node of the kinematization chain and click the **Attach** button from the **Kinematization** menu. **Result**: The kinematization chain is grouped within the container assembly and displayed as subnode of the **Assembly** node in the **Solution Explorer**.

Attaching a Kinematization Chain to an Assembly

To attach a kinematization chain that is available as a container assembly as described in Preparing a Kinematization Chain for Use as Kinematization Tool, page 76, proceed as follows:

Step	Action
1	Right-click an assembly, for example, a Lexium Cobot, in the **Model** view and run the command **Tool Manager > Attach Existing Tool > Use Existing Kinematization**. **Result**: The container assemblies that are available in the scene but not attached to an assembly are provided for selection.
2	Select the suitable container assembly **Assembly[n]** from the contextual menu. **Result**: A message is displayed requesting to you to decide whether you want to preserve the position and orientation of the container assembly with respect to the assembly. Proceed with step 3a clicking **Yes** or with step 3b by clicking **No**.
3a	Click **Yes** to preserve the position and orientation of the container assembly with respect to the assembly. **Result**: The container assembly is attached to the assembly without changing its position or orientation in the **Model** view but with an offset to the assembly. This offset is preserved and the container assembly is moved in accordance with the assembly changing position and or orientation within the scene.
3b	Click **No** to attach the container assembly to the assembly. **Result**: The container assembly is moved within the scene and attached to the assembly. Use the **Properties > Offsets Relative to Flange** parameters to adjust the orientation of the tool relative to the assembly.

A new node **Kinematized Tool - [n]** is displayed as subnode of the assembly, for example, the Lexium Cobot. The **Kinematized Tool - [n]** node provides the container assembly including the kinematization chain as subnode.

Managing Measurements Within the Model View

Overview

You can retrieve precise information about the position and orientation of body assemblies and joints that are available in the **Model** view by using the **Measurements Manager**. The position and orientation values are retrievable related to the global coordinate system or related to any other freely selectable object in the scene. With the plotting capability, the values of the position and orientation can be traced over time.

To access the function, right-click the body assembly in the **Model** view and run the command **Measurements Handler** from the contextual menu.

Commands of the Measurements Handler

Right-click the body assembly in the **Model** view to open the **Measurements Handler** contextual menu with the following commands:

Command		Description
Enable / Disable Measurements		Run this command to enable or disable the measuring function for the selected body assembly. By default, the function is disabled.
Measurements		The command is available after the function has been enabled with the **Enable Measurements** command. It allows you to retrieve position and orientation values of the selected body assembly relative to the global coordinate system or to another freely selectable object in the scene as well as plotting them to a graph with the following subcommands.
	Add New Measurement Relative To	Run the command and select the reference for the measurement: • **Global Coordinate System** • **[name of body assembly available in the Model view with measurements function enabled]**, for example, **Cylinder** **Result**: The **Measurand Selector** dialog box is displayed, page 79.
	Existing Measurements	After you have selected one or more measurands in the **Measurand Selector** dialog box, the selected measurands are available for selection with this subcommand and can be activated for plotting. Example: **Measurements > Existing Measurements > PositionX > Plot** **Result**: An individual **Measurand Plotter** view opens for each measurand that you select, page 80.
Export Multiple Measurements		The **Export Multiple Measurements** command allows you to export the values of measurands that were selected with the **Measurand Selector** dialog box to a .csv file. Run the command to open the **Multi Measurand Exporter** dialog box, page 81.

Measurand Selector Dialog Box

The **Measurand Selector** dialog box opens when you have selected the reference for the measurement with the **Measurements > Add New Measurement Relative To** command. It allows you to select the measurands that will be measured for the selected body assembly.

Select one or more of the following position or orientation measurands:

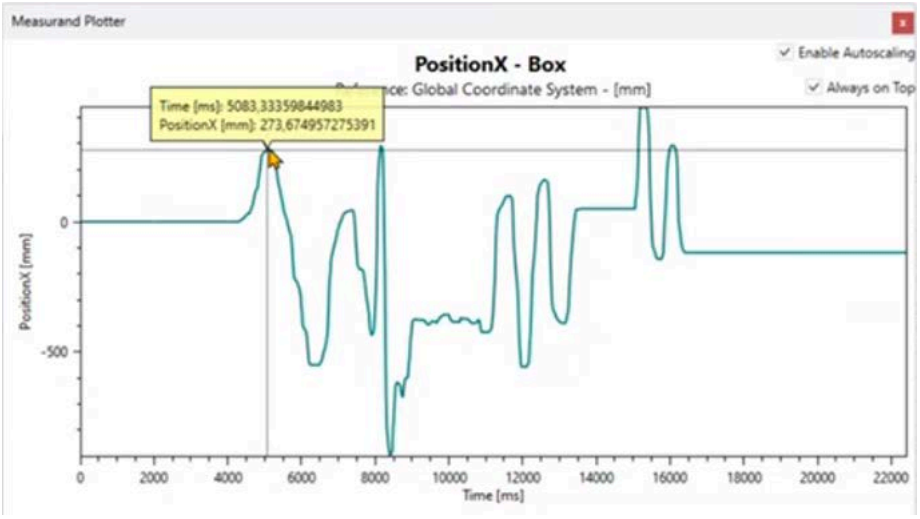
- **Position Measurands**: **PositionX**, **PositionY** or **PositionZ**
- **Orientation Measurands**: **RotX**, **RotY** or **RotZ**

Click **OK** to confirm the selection.

Measurand Plotter View

A **Measurand Plotter** view is displayed for each measurand selected with **Measurements > Existing Measurements > [position or orientation measurand] > Plot**

Each **Measurand Plotter** view displays the values of the selected measurand (for example, **PositionX** in mm or **RotationZ** in degrees) of the selected body assembly (for example, **Box**) related to the reference (for example, **Global Coordinate System**) over time in a coordinate system when the physical simulation is enabled and the scene is played.

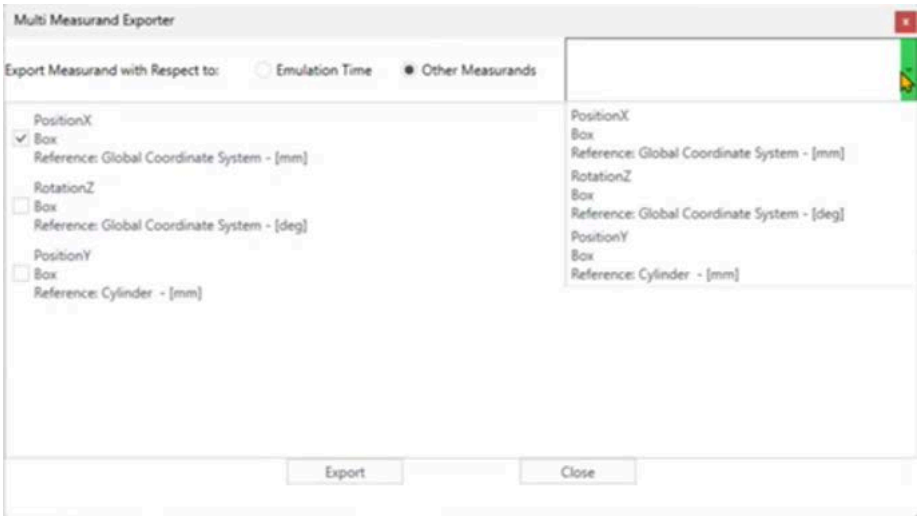


The following options are available:

Option	Description
Enable Autoscaling	By default, the **Enable Autoscaling** check box is selected and the graph is displayed within the given size of the **Measurand Plotter** view. When you scroll the mouse wheel to zoom in or out of the view, the **Enable Autoscaling** check box is automatically cleared and zooming in or out of the graph is performed at the cursor position. When you keep the right mouse button pressed and move the mouse up and down or left and right, you can move the complete graph within the window. To display the graph evenly within the **Measurand Plotter** view, select the **Enable Autoscaling** check box again.
Always on Top	By default, the **Always on Top** check box is selected to display the **Measurand Plotter** view in front of the EcoStruxure Machine Expert Twin window. When you clear the check box, the view is moved to the background when you click within the EcoStruxure Machine Expert Twin window.
Click a position within the graph	**Result**: The value of the measurand at the selected point in time is indicated in a yellow box.

Multi Measurand Exporter Dialog Box

Run the **Export Multiple Measurements** command to open the **Multi Measurand Exporter** dialog box. It lists the measurands that were selected with the **Measurand Selector** dialog box.



To select measurands for export to a .csv file, proceed as follows:

Step	Action
1	Select the check boxes of one or more measurands to export.
2	Select one of the **Export Measurand with Respect to:** options. • **Emulation Time** (default setting): Lists the measured values in the .csv file each with the respective time stamp. • **Other Measurands**: Open the list right to the option and select one of the measurands to create a .csv file that lists each measured value selected from the list below with reference to this measurand.
3	Click the **Export** button. **Result**: A **Save as** dialog box is displayed.
4	Browse to the folder where you want to save the .csv file, enter a **File name** and click the **Save** button. **Result**: The .csv file is created and saved to the selected folder.

Script View

What's in This Chapter

Script View

Overview

The **Script** view allows you to write scripts inside EcoStruxure Machine Expert Twin using the C# programming language.

To integrate the code in the program, click the **Build** button on the left side of the button bar or run the **Build project** command from the **Build** menu.

Errors that are detected during the build process are displayed in the **Error List** view below the **Script** view.

Also refer to What Can I Use the **Script** View For?, page 104.

Logs View

What's in This Chapter

Logs View

Overview

The **Logs** view provides information on how the program is running and displays different types of messages. By default, time stamps are added and the messages are color-coded according to their severity. You can disable the time stamp and the color in the Properties View, page 98.

NOTE: It is also possible to disable log messages completely. Thus, if the **Logs** view is empty, verify whether the **Mode** parameter in the **Properties** of the **Logs** view is set to **None**.

Message type	Description	Color
Action	Notification concerning a certain action executed in the scene.	black
Communication	Notification providing controller information, such as `connection set`.	black
Error	An error has been detected.	red
System	Information provided during start-up of EcoStruxure Machine Expert Twin.	blue
Debug	Customized messages defined in the **Script** view, page 82 with the command `'Log.DebugMessage<"Your message">'` to be generated, for example, when activating a specific object.	black
Warning	Advisory messages indicating that the scene is not playing properly, for example, due to the connection to the controller being interrupted.	orange
Information	General information about the instance.	green

Select the check box **Tail** to activate an automatic scrolling function to display new log messages that are added at the end of the **Logs** list. If this check box is not selected, you will have to scroll manually to the latest log messages at the end of the list.

Contextual Menu

Right-click a message to open a contextual menu with the following commands:

Command	Description
Clear	Deletes all messages from the **Logs** view.
Copy	Copies the selected line or lines to the clipboard.
Insert Separator	Inserts a line with the time stamp at the end of the messages. This can be useful to structure the log, for example, to mark the time when a test is being started.

Customizing the Logs View

To customize the log messages, select a message and configure your individual settings in the **Properties** view, page 98.

Loads View

What's in This Chapter

Loads View

Overview

The **Loads** view displays the loads that are available in the scene. This information is relevant for discrete events mode.

Selecting a load in the **Loads** view also selects the load in the **Model** view. Further information on the selected load is displayed in the **Properties** view, page 99.

Information Provided in the Loads View

To fill or update the columns of the **Loads** view, right-click within the **Loads** view and run the command **Refresh**. The following information is displayed:

Column	Description
Identification	Displays the identifier (usually a barcode) of each load. To modify the identifier of the load, edit the **Identification** parameter in the **Properties** view, page 99.
Next	Indicates the next action point the load will hit.
Destination	Indicates the final destination of the load.
Description	Provides further information on the load.

Further Information in the Properties View

For further information on the selected load and modification of the **Identification** parameter as well as the color, refer to the **Properties** view, page 99.

Inputs / Outputs Views

What's in This Chapter

Inputs / Outputs Views

Overview

Inputs and outputs (I/Os) are relevant for connections to controllers.

The **Inputs** view displays the inputs the controller receives, such as sensor signals.

The **Outputs** view displays the outputs that are sent by the controller, such as control commands.

Selected inputs / outputs are displayed in green.

Red cells within the table indicate that the parameter has not been configured correctly.

Contextual Menu

Right-click in the **Inputs** / **Outputs** table to open a contextual menu with the following commands:

Command	Description
Add	Adds a new input or output for the controller and adds a new row to this table. You can also add new inputs or outputs through components.
Refresh	Updates the content of the **Inputs** / **Outputs** table.
Export	Exports inputs or outputs to an Excel file.

Further Information in the Properties View

The **Properties** view allows you to modify the symbol and the description of the input / output.

Alarms View

What's in This Chapter

Alarms View

Overview

The **Alarms** view displays the alarms you have created for performing controller tests. To create an alarm, right-click a connection in the **Connections** view and run the **Alarms** command from the contextual menu to open the **Alarm Scheme** dialog box, page 68.

Information Provided in the Alarms View

The **Alarms** view provides the following information:

Column	Description
Connection	Indicates the controller to which the alarm belongs.
Alarm	Indicates the description that you have set for the alarm.

When an alarm is raised, it is displayed in the **Alarms** view, but will also be displayed as an advisory in the status bar of the **Model** view, page 74.

Joint Editor

What's in This Chapter

Joint Editor View

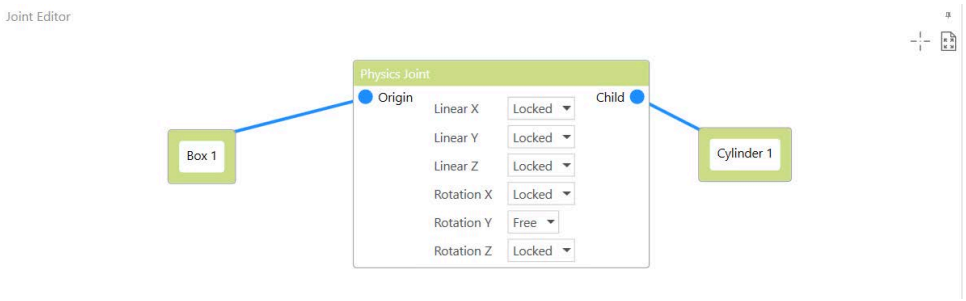
Overview

The **Joint Editor** view allows you to create passive and active **Physics Joints** between two or more body assemblies created with the **Kinematization** Menu, page 31.

The **Physics Joint** is configurable in the editor and allows you to lock, unlock or limit movement within the six degrees of freedom (DOF) individually.

For a joint, one reference is considered the stationary reference (rigid), while the other is considered the moving reference (physics). The moving and the stationary references of the joint must be located on two different body assemblies.

Example of a **Physics Joint** configuration in the **Joint Editor**:



The editor automatically verifies the configuration and indicates detected configuration issues by a red triangle in each box that is affected. A tooltip is provided that indicates a solution. The total number of issues detected in the editor is displayed in the lower left corner.

The **Joint Editor** provides two buttons in the upper right corner:

Button	Description
Moves the camera to position 0.0.	Click to move the camera view of the editor to the upper left edge.
Fits all nodes within the view	Click to zoom in or out of the editor view so that all available objects are displayed.

Creating Physical Joints

To create a physical joint between two assemblies, proceed as follows:

Step	Action
1	Right-click in the **Joint Editor** view and run the command **Add Body** from the contextual menu. **Result**: A new body block is displayed in the **Joint Editor**.
2	Create a relationship between the block in the **Joint Editor** and an assembly in the scene by selecting the assembly in the scene or in the **Solution Explorer**, right-clicking the block in the **Joint Editor** and running the command **Link to assembly** from the contextual menu. **Result**: The name of the assembly is assigned to the selected block in the **Joint Editor**.
3	Repeat steps 1 and 2 to create a second block in the **Joint Editor** and to link it to a second assembly in the scene. **Result**: Two blocks representing two different assemblies are available in the **Joint Editor**.
4	Right-click in the **Joint Editor** view and run the command **Add Physics Joint** from the contextual menu. **Result**: A **Physics Joint** block is added to the **Joint Editor**.
5	Click in the frame of the first assembly, hold down the left mouse button and draw a connecting line to the **Origin** connection point of the **Physics Joint** block. **Result**: The first assembly is defined as origin or stationary reference for the joint and the **Origin** connection point is highlighted in blue.
6	Verify the configuration of the body assembly that is connected as **Origin**: The **Dynamics** parameter **Type** must be set to **Rigid** or **Physics** (refer to the Properties of Body Assemblies, page 32). If the **Type** = **Bodiless** is selected, correct the configuration.
7	Click the **Child** connection point of the **Physics Joint** block, hold down the left mouse button and draw a connecting line to the second assembly. **Result**: The second assembly is defined as child or moving reference for the joint, the **Child** connection point is highlighted in blue.
8	Verify the configuration of the body assembly that is connected as **Child**: The **Dynamics** parameter **Type** must be set to **Physics** (refer to the Properties of Body Assemblies, page 32). If this is not the case, a red triangle is displayed at the upper right corner of the body block. To adapt the configuration, right-click the assembly that is defined as child and run the command **Set to Physics** from the contextual menu. **Result**: The **Dynamics** parameter **Type** is set to **Physics** and the red triangle is removed.
9	Configure the **Physics Joint**. For further information, refer to Configuring Physical Joints, page 90.

Configuring Physical Joints

By default, the six parameters representing the six degrees of freedom are set to **Axis Mode > Locked**, meaning that motion of the joint is not allowed:

- **Linear X**
- **Linear Y**
- **Linear Z**
- **Rotation X**
- **Rotation Y**
- **Rotation Z**

To allow motion in one or more directions, select **Axis Mode > Free** from the list for the respective parameters.

To restrict the motion of the moving reference for one degree of freedom, select **Axis Mode > Limited** and configure a force that corresponds to a spring to keep the body assembly within the defined limits.

The following constraints apply to the configuration of the **Axis Mode** with respect to the three axes and are restricted within the software:

Case	Axis 1	Axis 2	Axis 3	Description
Valid case 1	**Limited**: -75°...75°	**Limited**: -75°...75°	**Locked**	General rule: At least one axis must be **Locked** to help ensure stability. If one axis is **Locked**, allowed **Limits** for the other two axes are between -75° and 75°.
Valid case 2	**Limited**: -180°...180°	**Locked**	**Locked**	If two axes are **Locked**, the allowed **Limits** for third axis are between -180° and 180°.
Invalid case	**Limited**: -75°...75°	**Free**	**Locked**	This is an invalid case because if one axis is **Locked**. The software does not allow you set one of the other axes to **Free**.

With **Axis Mode > Limited** selected, configure the following parameters:

Parameter		Description
Limits		Configure limits for the motion of the moving reference.
	Min	Enter the maximum movement (in mm) in the negative direction for the remaining axis or axes if one or two axes are set to **Locked**. The valid range depends on the configuration of the **Axis Mode** for the three axes of motion as indicated in the table above: • -75°...75° with one axis set to **Locked** • -180°...180° with two axes set to **Locked**
	Max	Enter the maximum movement (in mm) in the positive direction for the remaining axis or axes if one or two axes are set to **Locked**. The valid range depends on the configuration of the **Axis Mode** for the three axes of motion as indicated in the table above: • -75°...75° with one axis set to **Locked** • -180°...180° with two axes set to **Locked**
Spring		Configure the physical properties that correspond to springs and that act on the moving reference to keep it within the configured limits.
	Stiffness	Enter a value >0 to configure the stiffness of the spring (in N/m) that is the extent to which it can resist deformation in response to an applied force.

Parameter		Description
	Damping	Enter a value >0 to configure the damping of the spring (in N·s/m) that is the resistance against fast changes in displacement and that contributes to bring the spring to rest quickly.
Enable Drive		Select this option to use active joints. Refer to Configuring Active Joints, page 91.

NOTE: In the **Properties** of the **Child** assembly, set the **Dynamics Parameters** to **Type** = **Physics** for allowing the physical simulation to display the motion in the scene.

By combining the physical joints feature with the functions provided in the **Kinematization** menu, you can create customized robot kinematics.

Configuring Active Joints

You can also configure active or motorized joints that interact with the geometry by using forces. To achieve this, set the **Axis Mode** to **Free** or **Limited** and configure the following parameters:

Parameter		Description
Enable Drive		Select this option to use active or motorized joints to keep a moving reference within the configured limits.
Drive Type		Select an option from the list to define the controller input that is valid for the motor: • **Velocity**: A velocity value for the motor (in **Units**) is provided by a variable from the controller. • **Forward/Backward**: Boolean forward/backward signals are provided by a variable from the controller. • **Custom**: Select this option to use customized motors or positioners. If they meet the requirements defined in Customized Motors or Positioners, page 39, they are available for selection in the **Custom Drive** list and can be added to the selected joint. You can configure them in the **Properties** view (see the *How to Use Device Catalogs User Guide*). The motor is configured with the parameter **Drive > Motor**.
Drive		Active joints are controlled by drives. The drive is a proportional derivative drive that applies a force according to the following formula: *F = stiffness * (target position - position) + damping * (target velocity - velocity)*
	Motor	Configure the motor according to the selection for the parameter **Drive Type**. The motor parameters are configured as described in the *How to Use Device Catalogs User Guide*. Alternatively, you can right-click the motor node in the **Solution Explorer** and control the motor manually with the **Stop**, **Start**, **Forward**, **Backward** commands.
	Spring	Configure the coefficients used by the spring drive to move the child body assembly (moving reference) through forces. Use the **Stiffness** parameter to apply a force that is proportional to the detected position error. Use the **Damping** parameter to apply a force that is proportional to the detected velocity error.

Logic Configurator

What's in This Chapter

Logic Configurator View

The **Logic Configurator** view allows you to configure logical relations between the elements of the scene in a graphical editor without the need for C# programming language skills. For further information, refer to the **Logic Configurator** User Guide.

Change History View

What's in This Chapter

Change History View

Overview

The **Change History** view records modifications performed in the **Model** view such as adding an object.

You can right-click an entry in the list and run the command **Set the assembly [ASSEMBLY NAME]'s properties back to what they were before this action** from the contextual menu to undo this specific action.

Solution Explorer View

What's in This Chapter

Solution Explorer View

Overview

The **Solution Explorer** view lists the components you are using in the **Model** in a tree structure consisting of categories:

- **Assemblies**
- **Motors**
- **Controllers**

By default, the items that are available under the **Assemblies** node are grouped by section, as first sub node and then by the type of assembly as second sub node (for example, Lexium P Robot). To avoid the categorization according to the type of assembly, click the button **Do not categorize the assemblies** in the upper part of the **Solution Explorer** view, on the left-hand side of the **Search** field.

Category Assemblies

The category **Assemblies** provides the **Sections** available in the **Model** as sub nodes. Assemblies selected in the **Solution Explorer** view are highlighted in the **Model** view. You can drag assemblies between different **Section** sub nodes. As an alternative, you can select the assembly you want to move and then change the section it is placed under in the **Properties** view. By editing the **Name** parameter in the **Properties** view, you can rename sections.

Right-clicking the **Assemblies** node opens a contextual menu with the following commands:

Command	Description
Add Section	Adds a node **Section[n+1]** below the last section node.
Show All Sections	Displays all sections that have been hidden previously.
Collapse All	Closes all sub nodes that are open.

Right-clicking a **Section** sub node opens a contextual menu with the following commands:

Command	Description
Hide All Sections But This	Displays in the **Model** view only the assemblies that are placed in the selected section.
Show All Sections	After running the command **Hide All Sections But This** this command restores the assemblies of all sections in the **Model** view.
Hide Assemblies	Hides the assemblies and the floor of the selected section from the **Model** view.
Lock Assemblies / **Unlock Assemblies**	Locks the assemblies that are available in the selected section. New assemblies that you are adding are not locked.
Fit	Resizes the floor to fit around the assemblies placed in the **Model** view.
Delete Section[n]	Deletes the selected section. It is not possible to delete the last remaining section.
Disable Assemblies	Disables the assemblies that are available in the selected section. To indicate this, they are displayed gray in the **Model** view. If you attempt to place a load on a disabled assembly, the load will “fall through” the assembly. To re-enable disabled assemblies, run the **Disable Assemblies** command again.
Collapse All	Closes all sub nodes that are open.

Right-clicking the sub node of an assembly allows you to disable or delete this assembly.

Category Motors

Right-clicking a motor sub node opens a contextual menu with the following commands:

Command	Description
Start / **Stop**	Starts / stops the motor.
Backward / **Forward**	Toggles the moving direction of the motor.
Copy	Copies the motor.
Disable	Disables the motor.
Delete	Deletes the motor.

Category Controllers

Right-clicking the **Controllers** node opens a contextual menu providing the **Load** command. It opens a **File Open** dialog box that allows you to browse for the controller file.

Your file contains script code created or modified by you or by someone else. The execution of the script code may compromise the security of your IT system.

⚠WARNING

INFORMATION SYSTEM VULNERABILITY

- Ensure the integrity of the script code contained in the file by ascertaining the originator and intent of the script before opening the file.
- Do not open the file if you cannot determine the originator and intent of the script, including confirming any modifications that may have been made.
- Verify and confirm that you want to execute this code when using the model contained in the file.

Failure to follow these instructions can result in death, serious injury, or equipment damage.

When attempting to open a file with a model containing script code, you are asked to acknowledge this advisory and accept responsibility for the execution of the code of the embedded script.

After the controller has been loaded, it will be displayed as a new sub node of the **Controllers** node.

Statistics View

What's in This Chapter

Statistics View

Overview

The **Statistics** view provides information about the loads in the scene monitored by sensors.

You can monitor different types of sensors, such as scanners, weights, counters and eaters. Scanners, for example, provide information about how many loads they have scanned. The information for buttons indicates how many times they have been pressed.

Managing Sensors

To add a sensor to the **Statistics** view, right-click it in the **Model** view and select the option **Observe** in the contextual menu.

To modify the name of a sensor, proceed as follows:

Step	Action
1	Select the sensor in the **Model** view.
2	Edit the **Name** parameter in the **Properties** view. **Result**: The name is changed in the **Model** view.
3	To display the new name also in the **Statistics** view, right-click the sensor in the **Model** view and clear the option **Observe** in the contextual menu and then select the option **Observe** again.

Properties View

What's in This Chapter

Properties View

Overview

The **Properties** view provides information about items selected in different views of EcoStruxure Machine Expert Twin, such as assemblies in the **Model** view, buttons in the **Control Panel** or **Connections**. Parameters of the **Properties** view that can be edited are displayed in black. Fixed values that cannot be edited are displayed in gray.

The buttons in the upper left part of the view allow you to select whether to display the properties as one list or grouped in categories.

Properties of the Logs View

To customize the log messages, select a message and configure your individual settings in the **Properties** view:

Message parameters:

Parameter	Description
Font	Enter the font type and size for displaying the selected log message type. For example: `[Font: Name=Courier New, Size=8.25, Units=3, GdiCharSet=0, GdiVerticalFont=False]`
Mode	Select from the list whether to display the messages in **Color** or in **Black**. For information on the types of messages and the corresponding color, refer to the **Logs** View, page 83. The option **None** switches off the log function. No new log messages will be displayed.
Time/Clock **Log Format**	Select the format of the time stamp from the **Time/Clock** list. The selected format is indicated in the **Log Format** line. • **None**: No time stamp is displayed for the log messages. • **Clock**: The time stamp is displayed in **Log Format**: `[HH:mm:ss.fff]` • **Elapsed**: The time stamp is displayed in **Log Format**: `[{0:dd\.hh\:mm\:ss\.fff}]` • **Simulated**: The time stamp is displayed in **Log Format**: `[{0:dd\.hh\:mm\:ss\.fff}]` • **DateTime**: The time stamp is extended by the date and displayed in **Log Format**: `[yyyy-MM-dd HH:mm:ss.fff]` **Elapsed** and **Simulated** differ if the time scale (parameter **Scale** in the **Model** menu, page 29) is set to value other than **1x** as the **Simulated** time would differ from elapsed time. The **Simulated** time is calculated from the **Elapsed** time multiplied by the selected scaling factor.
Search	Select the check box to highlight components that match a word in the selected log message.

File parameters:

If activated, log files are saved in the user directory of the logged in Windows user in the following default path:

%localappdata%\Schneider Electric\Machine Expert Twin\1\Work\Logs

Parameter	Description
Mode	The following options are provided for log files: • **None**: No log file is created. • **File**: A log file is created in the default path with the default name *debug.log*. • **FileDate**: A log file is created with the file name derived from the system date and time of day when EcoStruxure Machine Expert Twin has been started. • **FileModelName**: A log file is created when a project is opened or saved with the project file name as log file name.
LogFile Wrap Mode	This option is available if a log file is to be created. It defines how the log file is buffered: • **Continuous**: Append new log entries to an existing log file. • **Wrap**: Append new log entries to an existing log file until the maximum threshold is reached. Then the oldest entries are overwritten. • **Create new**: Append new log entries to an existing log file until the maximum threshold is reached. Then a new log file is created.

Filter parameters:

Parameter	Description
Text	Enter text you want to apply as a filter to the log messages. Only messages that contain this text are displayed in the **Logs** view.
Type	Select from the list the types of messages that are displayed in the **Logs** view: • **None** • **Action** • **Communication** • **Error** • **System** • **Debug** • **Warning** • **Information** For further information on the different message types, refer to the chapter **Logs** View, page 83.

Highlight parameters:

Parameter	Description
Text	Enter a text string you want to highlight in the log messages.
Color	Select the color to use for highlighting.

Properties of the Loads View

To customize a load, select it in the **Model** view and configure your individual settings in the **Properties** view:

In **Identification**, the **Type** parameter indicates the type of the load, for example, a box. With the **Identification** parameter, you can modify the identifier of the load which is usually a barcode.

With the other parameters, you can define physical characteristics of the load, the dimensions, the color and the position in space (in a cartesian coordinate system).

Properties of the Inputs / Outputs View

The **Properties** view allows you to modify the symbol and the description of the input or output.

Properties of the Alarms View

The **Properties** view provides further information on the alarm selected in the **Alarms** view, page 87 for controller tests.

Properties of the Solution Explorer View

The **Properties** view allows you to modify the settings of the **Assemblies**, **Motors** or **Controllers** selected in the **Solution Explorer** view. For further information on the properties of catalog objects, refer to the EcoStruxure Machine Expert Twin How to Use Device Catalogs User Guide.

Frequently Asked Questions

What's in This Part

FAQ

What's in This Chapter

Are Catalogs Saved With the EcoStruxure Machine Expert Twin Project?

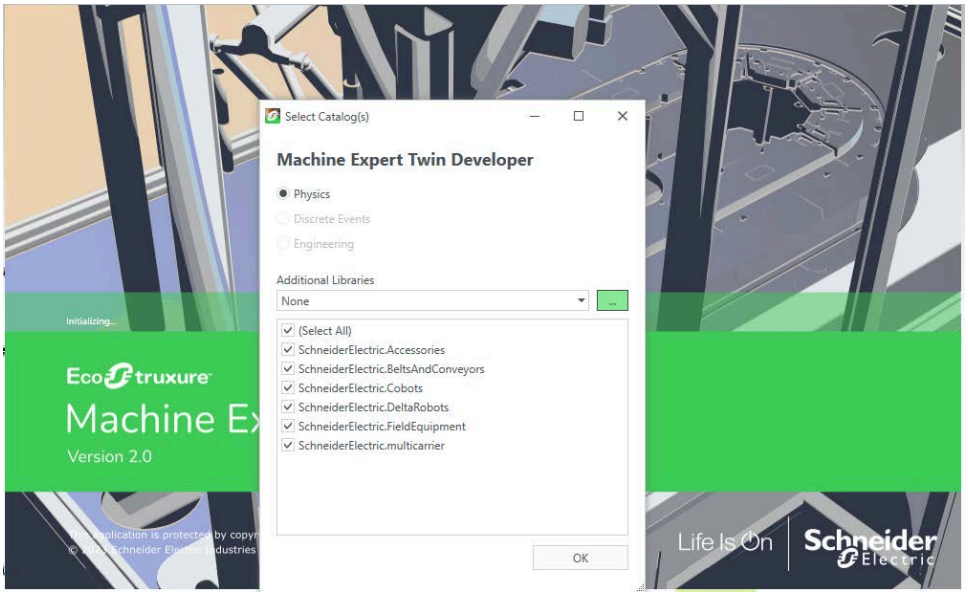
No, EcoStruxure Machine Expert Twin project files contain information about the scene, assemblies, properties, connections, linked controller input and output variables, camera views and scripts.

If you are using customized catalogs, references to the objects of the catalogs are stored in the EcoStruxure Machine Expert Twin project files.

How Can I Share a Project with Customized Catalogs?

When you open a EcoStruxure Machine Expert Twin project file, the catalogs referenced in the project must be available and are by default retrieved from the EcoStruxure Machine Expert Twin installation folder.

If you share a project that is using customized catalogs, provide the DLL files of the customized catalogs together with the project and save them on the target PC. Upon start-up of EcoStruxure Machine Expert Twin the **Select Catalog(s)** dialog box is displayed. Click the browse (**...**) button to browse to the folder where the customized catalogs are stored.



How Can I Position an Assembly Up And Down in the Scene?

If you select an assembly in the scene and move the mouse, the assembly is moved in the XY-plane.

To lock the position in the XY-plane and to move the assembly up and down in Z direction, hold down the **Shift** key and move the mouse.

How Can I Link or Snap Conveyors in the Scene?

To link two conveyors, hold down the **Ctrl** key and drag the red snapping point from one conveyor to the blue snapping point of another conveyor or vice versa. As soon as the snapping points collide, the assemblies are snapped and you can release the **Ctrl** key.

To unsnap two conveyors, select the component you want to unattach, hold down the **Ctrl** key and drag the component to another position in the scene.

How Can I Create a Random Load in the Scene?

Bring the cursor to the position where the load is to be created and press the **F** key to create a load of type box or hold down the **Shift** key and press the **F** key to create a load of type sphere.

To create a load on a conveyor, select the conveyor and press the **F** key.

How Can I Make the Camera View Following a Load?

Select a load and press the **F** key to make the camera following the load. To modify the zoom level, use the scroll wheel of the mouse while following the load.

To stop following a load, press the **F** key again.

What Can I Use the Script View For?

Intended Use

The **Script** view allows you to execute simple tasks inside a project such as performing clean-up actions on reset or automatically changing the camera view.

The **Script** view is not intended to be used for developing customized catalogs or assemblies.

In case you use scripts in your project, a message is displayed when the project is loaded.

Your file contains script code created or modified by you or by someone else. The execution of the script code may compromise the security of your IT system.

⚠WARNING

INFORMATION SYSTEM VULNERABILITY

- Ensure the integrity of the script code contained in the file by ascertaining the originator and intent of the script before opening the file.
- Do not open the file if you cannot determine the originator and intent of the script, including confirming any modifications that may have been made.
- Verify and confirm that you want to execute this code when using the model contained in the file.

Failure to follow these instructions can result in death, serious injury, or equipment damage.

When attempting to open a file with a model containing script code, you are asked to acknowledge this advisory and accept responsibility for the execution of the code of the embedded script.

Script Code Example

The following script code example modifies the camera view if the scene is locked.

```
using System;
using System.Numerics;
using System.Windows.Media;
using System.Linq;
using Experior;
using Experior.Core;
using Experior.Interfaces;
using Experior.Core.Loads;
using Experior.Core.Routes;
using Experior.Core.Communication.PLC;

public partial class Main
{
public int i = 0;
public int viewID =1;
public void Step(float deltatime)
{
//enable automatic change of camera view only when scene is
locked
if (Experior.Core.Environment.Scene.Locked)
{
// increment a count every 16ms (step time)
i = i + 1;
// check counter value 500 x 16ms = 8s
if (i >= 500)
{
// reset counter
```

```
i = 0;
//check is viewID is within valid limits and increment the
value
if (viewID < 9)viewID = viewID + 1;
else viewID = 1;
// change camera view with blending time of 1000ms
Experior.Core.Environment.Scene.Camera.MoveTo(viewID,1000);
// create logs message
Log.Write("Camera view changed to " + viewID.ToString());
}
}
}

public void Reset()
{
// reset camera view
Experior.Core.Environment.Scene.Camera.MoveTo(1,1000);
Log.Write("Camera view changed to 1");
}

    public void On(object trigger, string symbol)
{
}

public void Off(object trigger, string symbol)
{
}

}
```

How Can I Streamline the Start-up Process?

Overview

You can start EcoStruxure Machine Expert Twin with specific start-up options by using command-line arguments that you can enter from different locations:

- By using the Windows command prompt:

  Enter `cd C:\Program Files\Schneider Electric\EcoStruxure Machine Expert Twin` to change to the installation directory and append one of the command-line arguments listed below to `MachineExpertTwin.exe`
- By right-clicking the **Machine Expert Twin** icon on the desktop of your PC and running the **Properties** command. Select the **Shortcut** tab and append one of the command-line arguments listed below to the string in the **Target** field.

Command-Line Arguments

The following command-line arguments are available:

Command-line argument	Description
`-help` `-?`	Opens a command prompt with a list of available command-line arguments.
`-config`	Displays the **Select Catalog(s)** dialog box during start-up.
`-headless`	Starts EcoStruxure Machine Expert Twin in headless mode and enables the webapi and graphicsserver.

Command-line argument	Description
`-webapi [off|filepath]`	Enables webapi and specify the webapi configuration file to use or disable it (`[off]`). webapi is only accessible from localhost connections.
`-graphicsserver [off]`	Enables graphics server or disables graphics server `[off]`.
`-model <filepath>` `-modelfile <filepath>`	EcoStruxure Machine Expert Twin opens a project if the argument matches the name of an existing project.
`-phy` `-physics`	Starts EcoStruxure Machine Expert Twin in physics mode without displaying the **Select Catalog(s)** dialog box during start-up.
`—frictiontype <Patch|OneDirectional|TwoDirectional>`	Starts the physical simulation with the selected friction type (only available in physics mode). By default, EcoStruxure Machine Expert Twin uses the `Patch` mode.
`—libs <directory>`	Sets an additional library / catalog directory. Example: `-libs C:\Projects\Demo\` Two prerequisites need to be fulfilled: • The directory exists. • The directory contains an EcoStruxure Machine Expert Twin catalog DLL file.
`—debug [none|debug|detailed]`	Enables the debug level. For further information on the debug levels, refer to **Debug** Section, page 25.
`—autostart`	Starts the physical simulation after the project is loaded.

Glossary

D

digital twin:

A digital twin refers to a virtual representation or digital replica of a physical object, system, or process. It is a digital counterpart that simulates the behavior, characteristics, and performance of its physical counterpart in real-time or historical contexts. The concept of a digital twin allows for the integration of the physical and digital worlds, enabling organizations to monitor, analyze, and optimize the performance of their assets or processes.

EcoStruxure Machine Expert Twin provides features for visualization, simulation, and emulation of machines and automation lines throughout the complete lifecycle.

E

emulation:

Based on the *ISO 24765-2017 International Standard - Systems and software engineering--Vocabulary*, emulation is defined as the use of a data processing system to imitate another data processing system, so that the imitating system accepts the same data, executes the same programs, and achieves the same results as the imitated system.

M

Model view: In EcoStruxure Machine Expert Twin, the **Model** view provides the graphical representation of the scene.

P

physical simulation: The physical simulation is a software library that is designed to simulate and model physical systems in a computer-generated environment. It is used to create realistic and dynamic animations and simulations of objects, environments, and interactions between them. In EcoStruxure Machine Expert Twin the physical simulation uses mathematical algorithms to simulate physical phenomena, such as gravity, friction, and collision detection.

project: An EcoStruxure Machine Expert Twin project file is saved with the extension **.experior*. It contains the information about assemblies, connections, loads, settings.

S

scene: In the EcoStruxure Machine Expert Twin context, a scene is a representation of a set of assemblies interacting with loads.

simulation:

Based on the *ISO 24765-2017 International Standard - Systems and software engineering--Vocabulary*, simulation describes two concepts:

- A model that behaves or operates like a given system when provided a set of controlled inputs.
- The use of a data processing system to represent selected behavioral characteristics of a physical or abstract system.

In the context of this manual, the term simulation is used whenever it is referred to modeling physical systems in EcoStruxure Machine Expert Twin.

STEP: (STandard for the Exchange of Product model data) This ISO 10303 standard specifies a standard file format for 3-D models and allows for product data exchange across different platforms.

U

URDF: (unified robotics description format) A special type of eXtensible Markup Language (XML) file that includes the physical description of a robot and contains information on the mechanical structure, joints, 3-D modelling graphics, motors and colliders. URDF files are provided by numerous robotic manufacturers for download. EcoStruxure Machine Expert Twin allows importing URDF files for integrating third-party robots into a project without manual programming.

Index

P

R

S

T

U

V

W

X

Z

Schneider Electric
35 rue Joseph Monier
92500 Rueil Malmaison
France

+ 33 (0) 1 41 29 70 00

www.se.com

As standards, specifications, and design change from time to time, please ask for confirmation of the information given in this publication.

EIO0000005022.04