UNIVERSAL ROBOTS

Software Handbook

PolyScope X

Original instructions (en)

SW 10.12.1

Contents

1. Robot Arm Basics

Robot Arm

The Universal Robots robot arm is composed of tubes and joints. The coordinated motion of these tubes and joints, via PolyScope X software, moves the robot arm.

- **Base**: where the robot arm is mounted.
- **Shoulder** and **Elbow**: where the larger movements originate.
- **Wrist 1** and **Wrist 2**: where the finer movements originate.
- **Wrist 3**: where the tool attaches to the tool flange.

You can attach a tool to the flange at the end of Wrist 3. Moving the robot arm positions the tool.

> **CAUTION**
> You cannot position the tool directly above, or directly below the Base.

Teach Pendant

The Teach Pendant, the touch screen that controls the robot, is optimized for use in industrial environments. Unlike consumer electronics, the Teach Pendant touch screen sensitivity is, by design, more resistant to environmental factors such as:

- Water droplets and/or machine coolant droplets
- Radio wave emissions
- Other conducted noise from the operating environment

The touch sensitivity is designed to avoid false selections on the interface, and to prevent unexpected motion of the robot.

Using the screen

For best results, use the tip of your finger to make a selection on the screen.
In this manual, this is referred to as a **tap**.
A commercially available stylus may be used to make selections on the screen if desired.

2. Installing the Robot

Assembling the Robot Arm and Control Box

To start using PolyScope X, make sure your robot arm and Control Box are assembled and the power cable is plugged in.
If the robot is not assembled, you may need to assemble and mount the robot arm and Control Box.

> **WARNING**
> Tipping hazard. If the robot is not securely placed on a sturdy surface, the robot can fall over and cause injury.

To assemble and power-on the robot arm

1. Unpack the robot arm and the Control Box.
2. Mount the robot arm on a sturdy, vibration-free surface, using screws and a hex key (Allen wrench).

   Mounting the robot may require two people.
3. Place the Control Box on its Foot.
4. Connect the robot cable to the robot arm and the Control Box.
5. Plug in the main/power cable of the control box.
6. Press the power button on the Teach Pendant to turn on the robot.

2.1. Robot Arm

Description

The robot arm, depending upon weight, can be carried by one or two people unless the sling is provided. If the sling is provided, equipment for lifting and transport is required.

2.2. Control Box and Teach Pendant

Description

The Control Box and the Teach Pendant can each be carried by one person.
While in use, all cables are to be coiled and held to prevent tripping hazards.

2.3. Assembly and Mounting

Description Install and power on the robot arm and Control Box to start using PolyScope.

Assemble the robot You have to assemble the robot arm, Control Box and Teach Pendant to be able to continue.

1. Unpack the robot arm and the Control Box.
2. Mount the robot arm on a sturdy, vibration-free surface.
3. Place the Control Box on its Foot.
4. Connect the robot cable to the robot arm and the Control Box.
5. Plug in the mains, or main power cable, of the Control Box.

WARNING

Failure to secure the robot arm to a sturdy surface can lead to injury caused by the robot falling.

- Ensure the robot arm is secured to a sturdy surface

3. 3PE Teach Pendant Installation

Description

The 3-Position Enabling Teach Pendant (3PE TP) is a safety-critical interface designed to enhance manual control. Integrated directly into the Teach Pendant, the 3PE buttons ensure that robot motion can only be initiated when the operator maintains a controlled grip.

3.1. Hardware Installation

after demo, verify this content

To remove a Teach Pendant

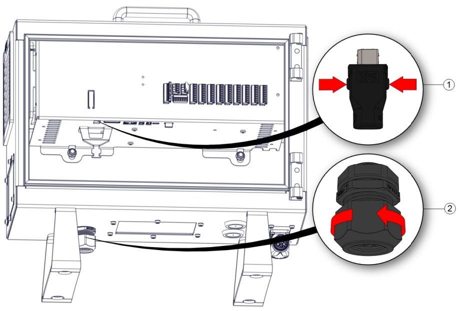
NOTICE

Replacing the Teach Pendant can result in the system reporting a fault on start-up.

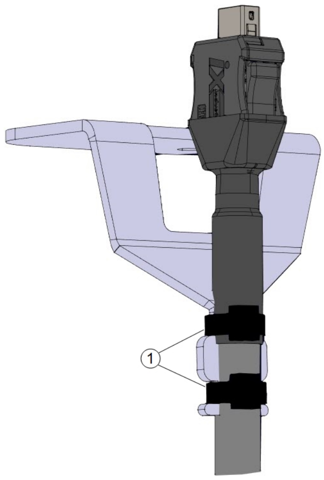
- Always select the correct configuration for the type of Teach Pendant.

To remove the standard Teach Pendant:

1. Power down the control box and disconnect the main power cable from the power source.
2. Remove and discard the two cable ties used for mounting the Teach Pendant cables.
3. Press in the clips on both sides of the Teach Pendant plug as illustrated, and pull down to disconnect from the Teach Pendant port.
4. Fully open/loosen the plastic grommet at the bottom of the control box and remove the Teach Pendant plug and cable.
5. Gently remove the Teach Pendant cable and Teach Pendant.



1	Clips	2	Plastic grommet



1	Cable ties

To install a 3PE Teach Pendant

1. Place the Teach Pendant plug and cable in through the bottom of the control box and fully close/tighten the plastic grommet.
2. Push the Teach Pendant plug into the Teach Pendant port to connect.
3. Use two new cable ties to mount the Teach Pendant cables.
4. Connect the main power cable to the power source and power on the control box.

There is always a length of cable with the Teach Pendant that can present a tripping hazard if it is not stored properly.

- Always store the Teach Pendant and the cable properly to avoid tripping hazards.

3.1.1. Teach Pendant with 3-Position Enabling (3PE) Device

Description

Depending on the robot generation, your Teach Pendant can include a built-in 3PE device. This is called a 3-Position Enabling Teach Pendant (3PE TP).
Higher payload robots can only use the 3PE TP.

If you are using a 3PE TP, the buttons are located on the underside of the Teach Pendant, as illustrated below. You can use either button, according to your preference.

If the Teach Pendant is disconnected, you are required to connect and configure an external 3PE device. The 3PE TP functionality extends to the PolyScope interface, where there are additional functions in the Header.

> **NOTICE**
>
> - If you have bought a UR15, UR20 or UR30 robot, a Teach Pendant without the 3PE device will not work.
> - Using a UR15, UR20, or UR30 robot requires an external enabling device or a 3PE Teach Pendant when programming, or teaching, within the reach of the robot application. See ISO 10218-2.
> - The 3PE Teach Pendant is not included with the purchase of the OEM Control Box, so enabling device functionality is not provided.

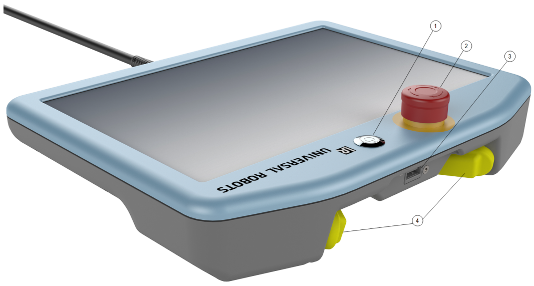
Robot and 3PE TP Compatibility Table

Compatibility Table per UR Robot Generations		
Robot Generation	**Standard Teach Pendant**	**3PE Teach Pendant**
e-Series	√	√
UR Series		√
Please note that UR Series robots are not compatible with the Standard Teach Pendant.		

Compatibility Table per Individual UR Robots			
Robot Model*	**Standard Teach Pendant**	**3PE Teach Pendant**	**No Teach Pendant**
UR3e	√	√	√
UR5e	√	√	√
UR7e	√	√	√
UR10e	√	√	√
UR12e	√	√	√
UR16e	√	√	√
UR8 Long		√	√
UR15		√	√
UR18		√	√
UR20		√	√
UR30		√	√
**This includes both Standard and OEM Control Box.*			

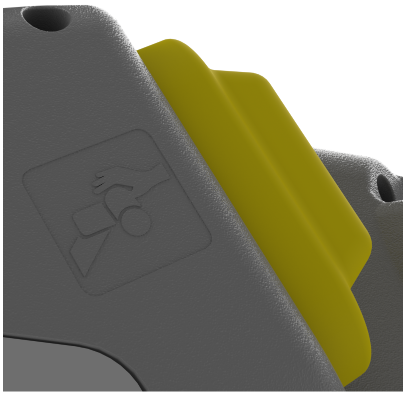
Overview of TP

1. Power button
2. Emergency Stop button
3. USB port (comes with a dust cover)
4. 3PE buttons



Freedrive

A Freedrive robot symbol is located under each 3PE button, as illustrated below.



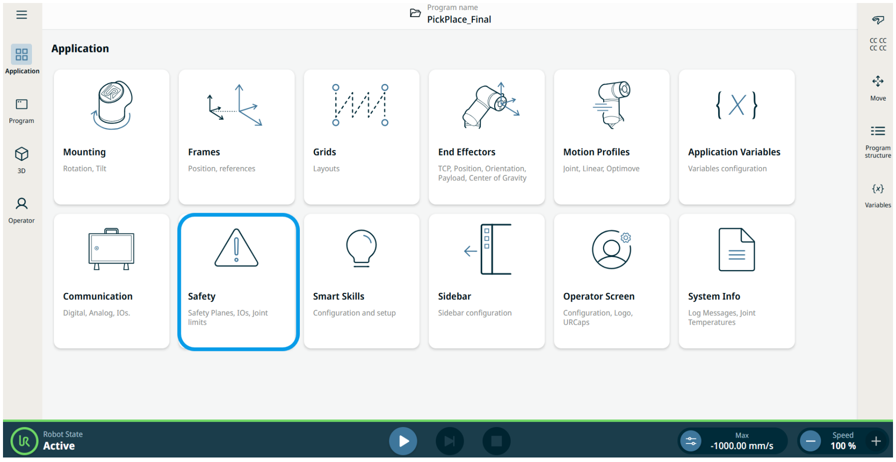
Further 3PE TP details

The 3PE device = **yellow hardware buttons** on the Teach Pendant.
The Polyscope on the 3PE TP is always in Teach Pendant 3PE **Stop mode**.
To enter **Normal mode**, press a 3PE device halfway down and hold.
To enter **Freedrive mode**, double-click a 3PE device halfway down and hold.

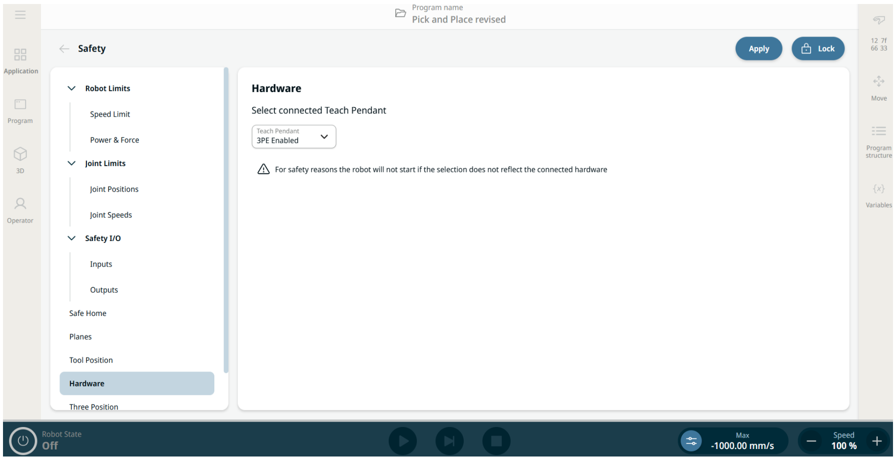
3.2. Software Installation

To configure the 3PE TP software

1. On PolyScope, in the left menu, tap **Application** and select **Safety**.



2. Tap **Hardware** and the **Unlock** button.



3. Enter the password and tap **Confirm**. Teach Pendant is now enable.
4. Tap **Apply** to restart the system. PolyScope continues to run.
5. Tap **Apply and restart** then **Confirm Configuration** to complete the 3PE Teach Pendant software installation.

4. First Boot

Description

The first boot is the initial sequence of actions you can take with the robot after assembly. This initial sequence requires you to:

- Power on the robot
- Insert the serial number
- Initialize the robot arm
- Power down the robot

CAUTION

Failure to verify the payload and installation before starting up the robot arm can lead to injury to personnel and/or property damage.

- Always verify the actual payload and installation are correct before starting up the robot arm.

CAUTION

Incorrect payload and installation settings prevent the robot arm and Control Box in functioning correctly.

- Always verify the payload and installation setting are correct.

NOTICE

Starting up the robot in lower temperature can result in lower performance, or stops, due to temperature-dependent oil and grease viscosity.

- Starting up the robot in low temperatures can require a warmup phase.

4.1. Powering On the Robot

To power on the robot

Powering on the robot turns on the Control Box and loads the display on the TP screen.

1. Press the power button on the Teach Pendant to power on the robot.

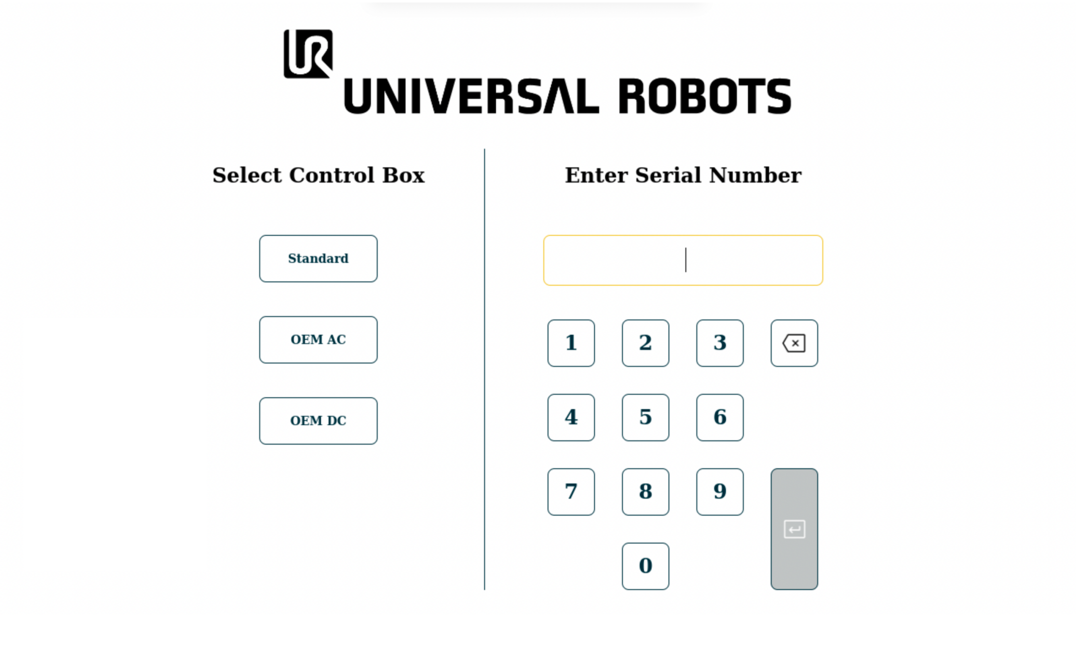
4.2. Inserting the Serial Number

To insert the serial number

Installing your robot for the first time requires you to enter the serial number on the robot arm. This procedure is also required when you re-install the software. For example, when you install a software update.

1. Select your Control Box.
2. Add the serial number as it is written on the robot arm.
3. Tap **OK** to end.

It can take a few minutes for the start screen to load.

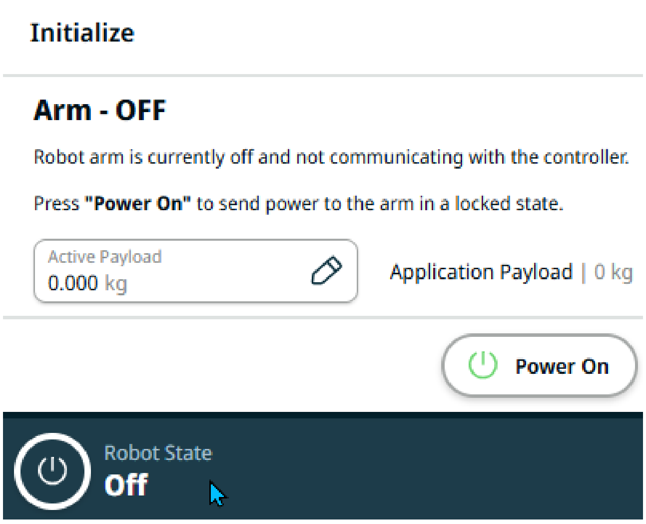


4.3. Starting the Robot Arm

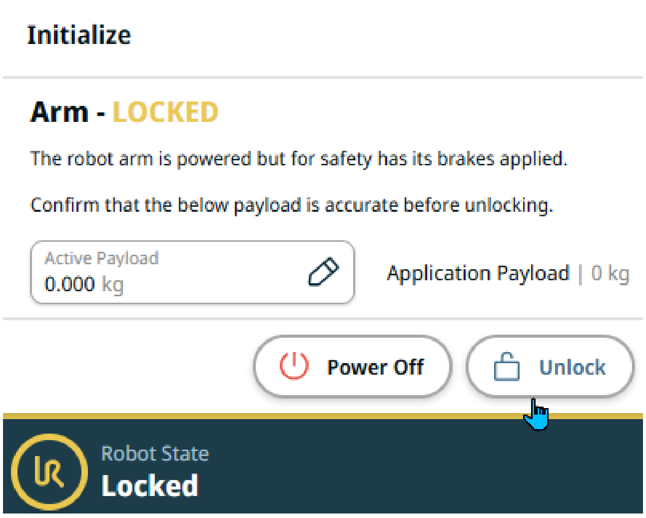
To start the robot

Starting the robot arm disengages the braking system, allowing you to start moving the robot arm and to start using PolyScope X.

1. In the left side of the footer, tap the power button or **Robot State** icon. The robot arm state is **Off**.
2. When the Initialize box displays, tap **Power On**. The robot arm state is **Booting**.

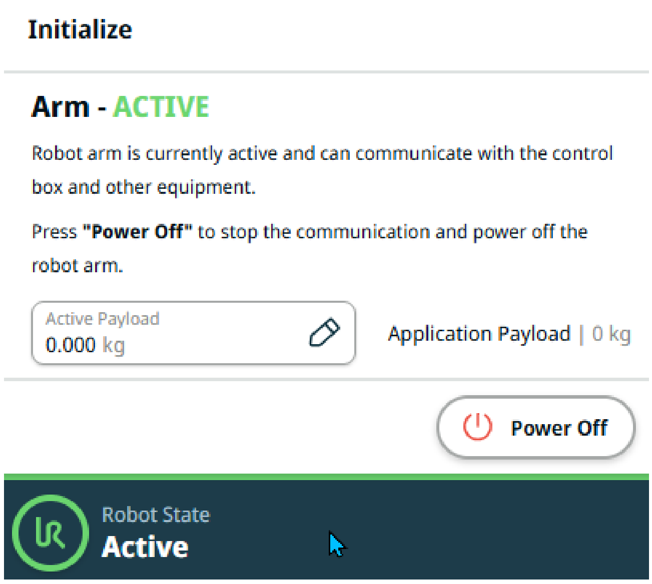


3. Tap **Unlock** to release the brakes.



Robot arm initialization is accompanied by sound and slight movements as the joint brakes are released.

4. The robot arm state is now **Active**, and you can start to use the interface.



5. You can tap **Power Off** to turn off the robot arm.

When the robot arm state changes from **Idle** to **Normal**, sensor data is checked against the configured mounting of the robot arm.
If the mounting is verified, tap **START** to continue releasing all joint brakes, preparing the robot arm for operation.

4.4. Powering Down the Robot

To power down the robot arm

> **WARNING**
>
> Unexpected start-up and/or movement can lead to injury
>
> - Power down the robot arm to prevent unexpected start-up during mounting and dismounting.

1. At the left side of the footer, tap the **Robot State** icon to turn off the robot arm.

   The icon color changes from green to white.
2. Press the power button on the Teach Pendant to turn off the Control Box.
3. If a Shutdown dialog box displays, tap **Power Off**.

At this point, you can continue to:

- Unplug the mains cable / power cord from the wall socket.
- Allow 30 seconds for the robot to discharge any stored energy.

4.5. Starting the Robot Arm

WARNING
Always verify the actual payload and installation are correct before starting up the robot arm. If these settings are incorrect, the robot arm and Control Box will not function correctly and may become dangerous to people or equipment.

CAUTION
Ensure the robot arm is not touching an object (e.g., a table) because a collision between the robot arm and an obstacle might damage a joint gearbox.

To start the robot:

- Tap the **Robot State** icon in the footer, followed by the green icon **Power On** button to start the initialization process.

  The **Robot State** icon turns yellow and states **Locked** to indicate the power is on and in Idle. The **Power Off** and **Unlock** buttons appear.

  An **Active Payload** field is available to input data in kilogram.
- Tap the yellow **Unlock** button to release the brakes.
- Tap the red **Power Off** to power off the robot arm.

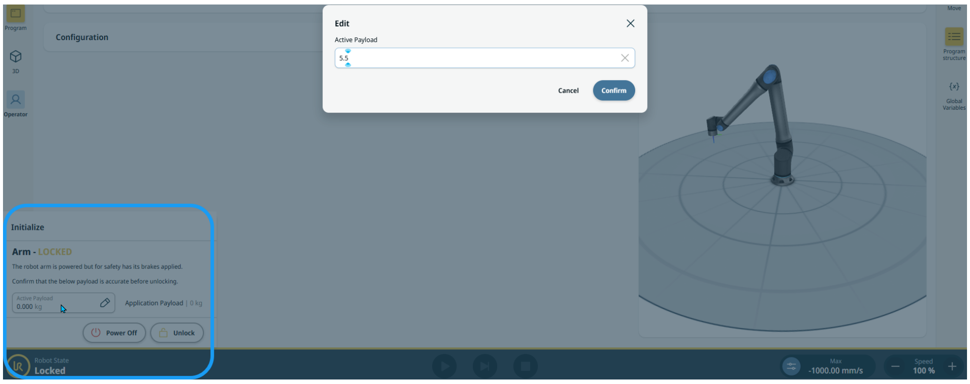
4.6. Safely Setting the Active Payload

Verify installation

Before using PolyScope X, verify that the Robot Arm and Control Box are correctly installed.

1. On the Teach Pendant, press the emergency stop button.
2. On the screen, tap **OK** when the Robot Emergency Stop box appears.
3. On the Teach Pendant, press the power button and allow the system to start and load PolyScope X.
4. Tap the on-screen **Power** button at the bottom left of the screen.
5. Hold and twist the emergency stop button to unlock.
6. On the screen's footer, verify the **Robot State** is **Off**.
7. Step outside the reach (workspace) of the robot arm.
8. Tap the on-screen **Power** button.
9. In the Initialize box, tap **Power On**, and the robot state is changed to **Locked**.
10. In the Active Payload, verify the payload mass.

    You can also verify the mounting position is correct, in the 3D view.
11. Tap the **Active Payload** field, and an **Edit** field appears in the main screen.
12. Enter your active payload and **Confirm**.



13. Tap **Unlock** for the robot arm to release its brake system.

4.7. Freedrive

Description

Freedrive lets you manually pull the robot arm into position.
On most robot sizes, enable Freedrive by pressing the **Freedrive** button on the Teach Pendant. Additional ways to enable and use Freedrive are described in the following sections.
In Freedrive, the robot arm joints move with little resistance because the brakes are released. Resistance increases as the robot arm approaches a predefined limit or plane, so pulling the robot into position feels heavier near those boundaries.

> **WARNING**
> Injury to personnel can occur due to unexpected motion.
> - Verify the configured payload matches the payload in use.
> - Verify the correct payload is securely attached to the tool flange.

Enabling Freedrive

You can enable Freedrive in the following ways:

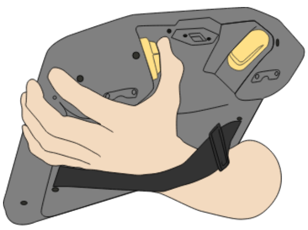
- Use the 3PE Teach Pendant
- Use the Freedrive on robot
- Use I/O actions

> **NOTICE**
> Enabling Freedrive while you are moving the robot arm can cause drift and faults.
> - Do not enable Freedrive while you are pushing or touching the robot.

3PE Teach Pendant

Use the 3PE TP button to put the robot arm in Freedrive:

1. Quickly light-press, release, then light-press again and hold the 3PE button in that position.



Now you can pull the robot arm into a desired position, while the light-press is maintained.

Freedrive on robot

Use **Freedrive on robot** to move the robot arm:

1. Press and hold the switch configured for **Freedrive on robot**.
2. When the **Freedrive** panel appears in PolyScope, select the movement type for the robot arm joints, or use the list of axes to customize the movement type.
3. If needed, define the feature type by selecting an option from the **Feature** dropdown list.

   The robot arm can stop moving if it approaches a singularity. Tap **All axes are free** in the **Freedrive** panel to resume movement.
4. Move the robot arm as desired.

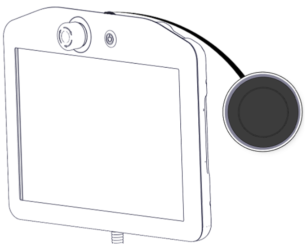
Backdrive

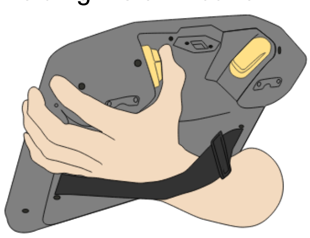
During initialization of the robot arm, minor vibrations can occur when the robot brakes release. In situations such as when the robot is close to a collision, these vibrations are undesirable. Use Backdrive to force specific joints into position without releasing all brakes in the robot arm.

To test Freedrive

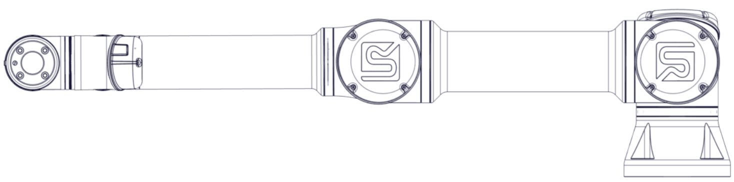
1. Unmount attachment or set TCP (tool center point)/payload/CoG (center of gravity) according to tool specifications.
2. To move the robot arm in freedrive, see figure below.

Standard TP	3PE TP
On a standard teach pendant (TP), press and hold the freedrive button.	On a 3PE TP, rapidly light-press, release, light-press again, and keep holding the 3PE button in this position.





3. Pull/Push the robot to a horizontally elongated position and release. See figure below.



Elongated horizontal position of UR robot

4. Verify the robot arm can maintain the position without support and without pressing the freedrive or 3PE button.

4.7.1. Freedrive Input

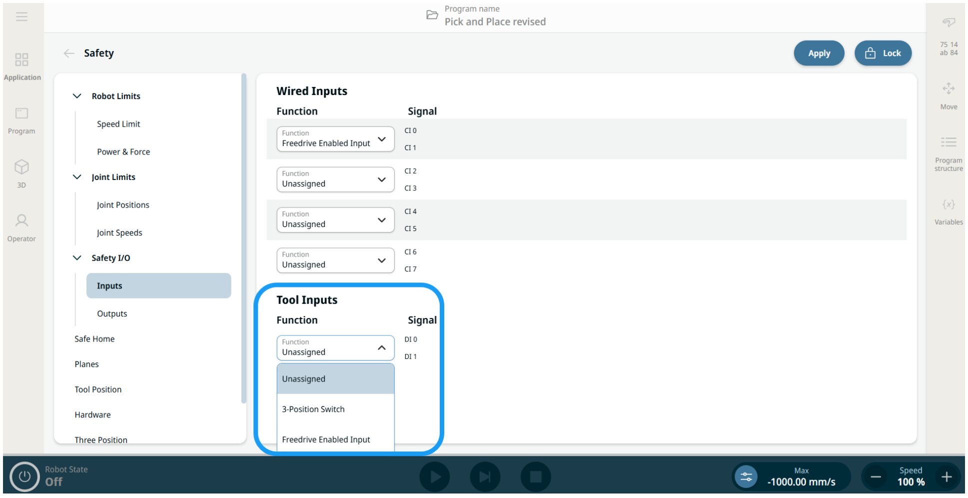
Description

Freedrive input is recommended to be used with 3PE TP and/or 3 Position Enabling Device input.
When Freedrive **INPUT** is `High`, the robot will only enter Freedrive if the following conditions are satisfied:

- 3PE TP button is not pressed
- 3 Position Enabling Device **INPUT** either not configured or not pressed (**INPUT** `Low`)

Freedrive Enabled Input

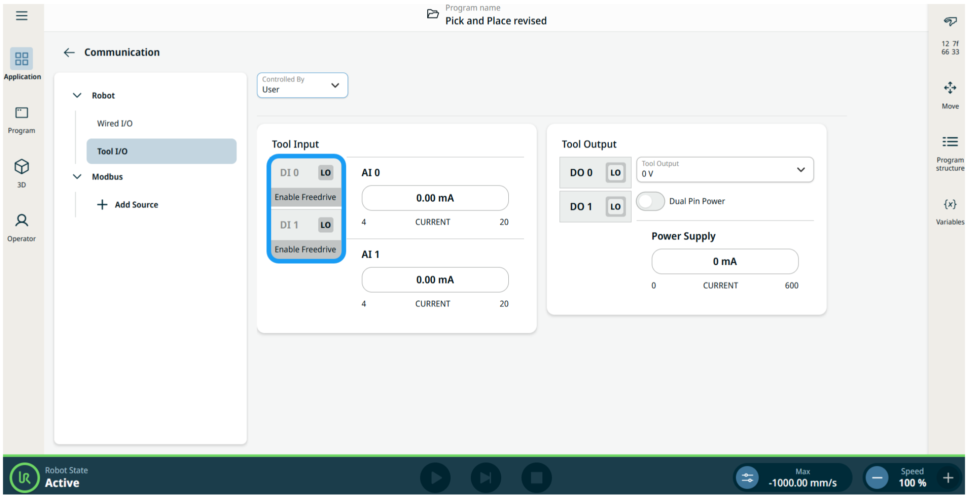
On UR8 Long robots, the **Freedrive Enabled Input** and **3-Position Switch** safety functions can be assigned to the **Tool Inputs** on the application **Safety** screen.



NOTICE

This functionality is currently only available on the UR8 Long robots. On all other robot types, the new Tool Inputs section is not shown on the Safety I/O screen.

When the Tool inputs are assigned a Safety Function, it is not possible to rename a Tool input or assign a Tool input an action on the Communication **Tool IO** screen.



5. Installation

Description

Installing the robot can require the configuration and use of input and output signals (I/Os). These different types of I/Os and their uses are described in the following sections.

5.1. Ethernet

Description

The Ethernet interface can be used for:

- MODBUS, EtherNet/IP and PROFINET.
- Remote access and control.

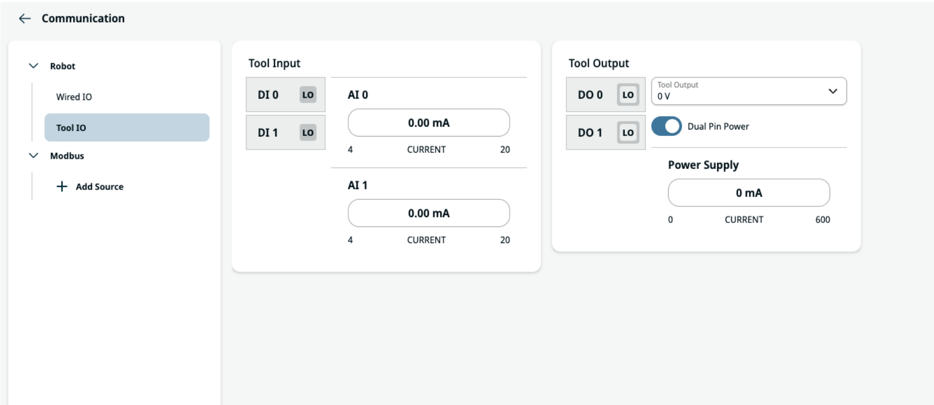
To connect the Ethernet cable by passing it through the hole at the base of the Control Box, and plugging it into the Ethernet port on the underside of the bracket.
Replace the cap at the base of the Control Box with an appropriate cable gland to connect the cable to the Ethernet port.

The electrical specifications are shown in the table below.

Parameter	Min	Typ	Max	Unit
Communication speed	10	-	1000	Mb/s

5.2. Input and Output

Description The tool IO tab monitors and sets the live IO signals from/to the Control Box.



Tool Output The tool output contains two digital output fields, tool output voltage and power supply current indicator and Dual Pin Power toggle.

- **Digital Output (DO)** - can be independently set to high or low
- **Tool Output Voltage** - selectable 0V, 12V and 24V. This setting is persistent over restarts of the robot controller
- **Power Supply** - current consumption indicator
- **Dual Pin Power** - used to toggle between digital outputs and source of power for the tool. Enabling Dual Pin Power disables the default tool digital outputs (DO)

After selecting a new output configuration, the changes take effect. The currently loaded installation is modified to reflect the new configuration. After verifying the tool outputs are working as intended, make sure to save the installation to prevent losing changes.

Tool Input The tool input contains two digital input (DI) fields and two fields for analog input (AI) currents.

DI fields can be independently configured to action presets:

- Start Program
- Stop Program
- Pause Program
- Freedrive

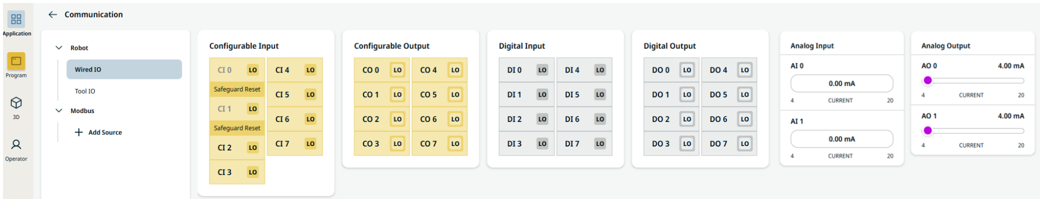
The AI's can be set from 4-20mA. These settings are persistent over restarts of the robot controller and saved in the installation.

5.2.1. Using the Wired I/O Tab

Description

Use the Wired I/O Tab screen to monitor and set the live I/O signals from/to the Control Box.

The screen displays the current state of the I/O, including during program execution. The program stops if anything is changed during execution. At program stop, all output signals retain their states. The screen updates at 10Hz, so a very fast signal might not display properly.



Configurable I/Os

Configurable I/Os can be reserved for special safety settings defined in the I/O Setup. Under those which are reserved will have the name of the safety function in place of the default or user defined name.

Configurable inputs that are reserved for safety settings are not togglable and will be displayed as LED's only.

For unreserved I/Os, it has the following options:

- Start Program
- Stop Program
- Pause Program
- Freedrive

Digital I/Os

DI have the following options:

- Start Program
- Stop Program
- Pause Program
- Freedrive

All DIs are preset to Low.
All DOs are set independently to either high or low.

Analog IOs

The analog I/O's can be set to either current [4-20mA] or voltage [0-10V] output. These settings are persistent over restarts of the robot controller and saved in the installation.

5.2.2. Drive Power Indicator

Description

The drive power indicator is a light that turns on when the robot arm is powered on, or when there is power to the robot cable. When the robot arm is powered off, the drive power indicator turns off.

The drive power indicator is connected via the Digital Outputs. It is not a safety feature and does not use safety I/Os.

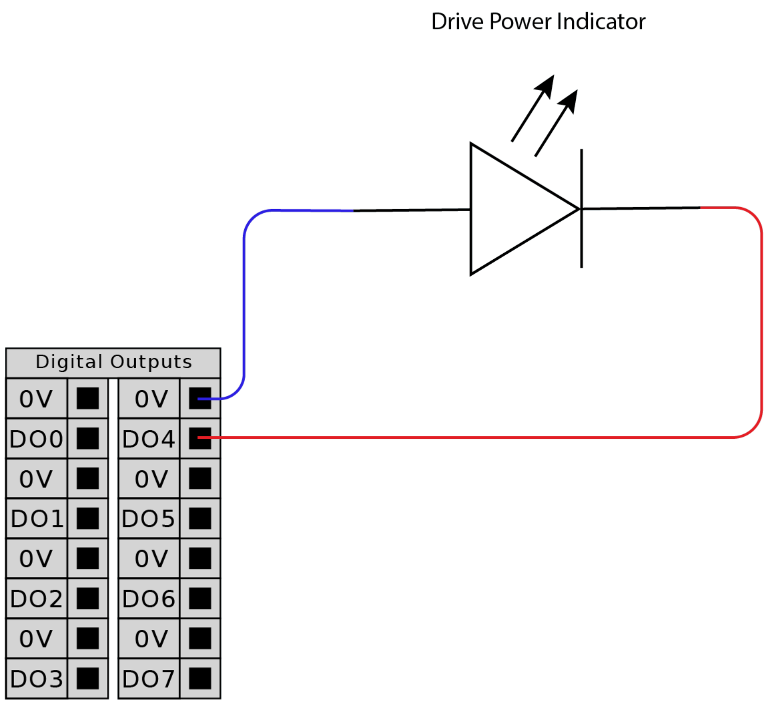
Indicator

The drive power indicator can be a light that can work at 24VDC.

To set up the indicator

Setting up the indicator requires a light and wiring for the outputs.

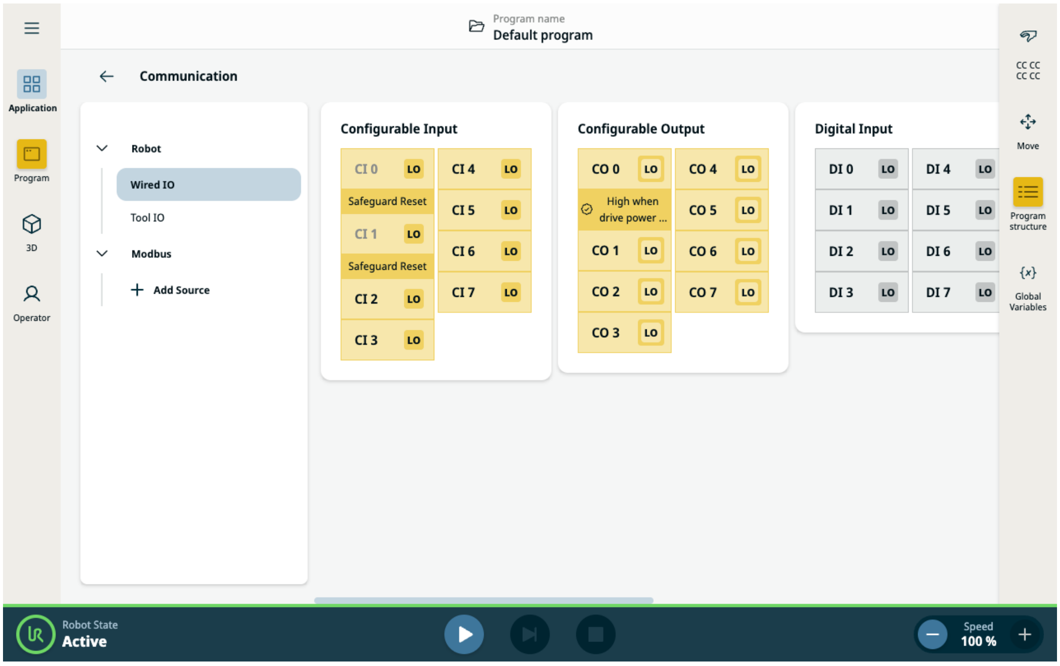
1. Connect your drive power indicator to the Digital Outputs as shown in the image below.
2. Verify the drive power indicator is correctly connected.
   - You can power on the robot arm and verify the light turns on.
   - You can power off the robot arm and verify the light turns off.

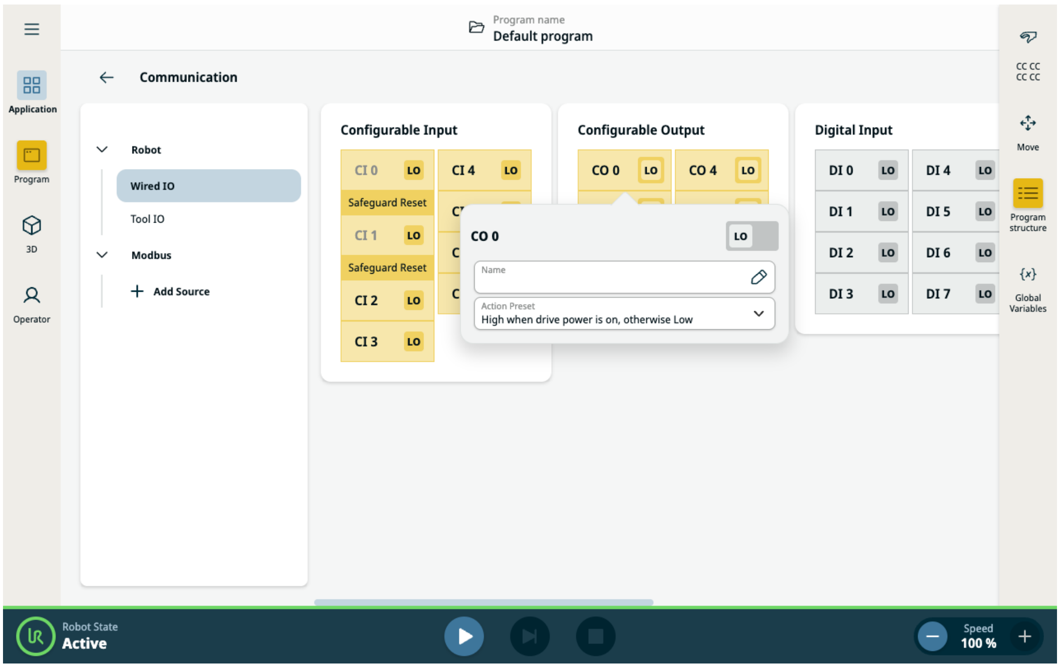


To configure the indicator

1. In the Navigation menu, tap **Application**.
2. Select **Communication**.
3. In the side menu, select **Wired IO**.
4. Scroll to your desired output type and tap to select one of the following:
   - Configurable Output
   - Digital Output
   - Analog Output
5. Select **Action Preset**

   You can name the selected output
6. In the dropdown select **High when drive power is on, otherwise Low**.





5.2.3. Remote Mode in Safety Overview

Description

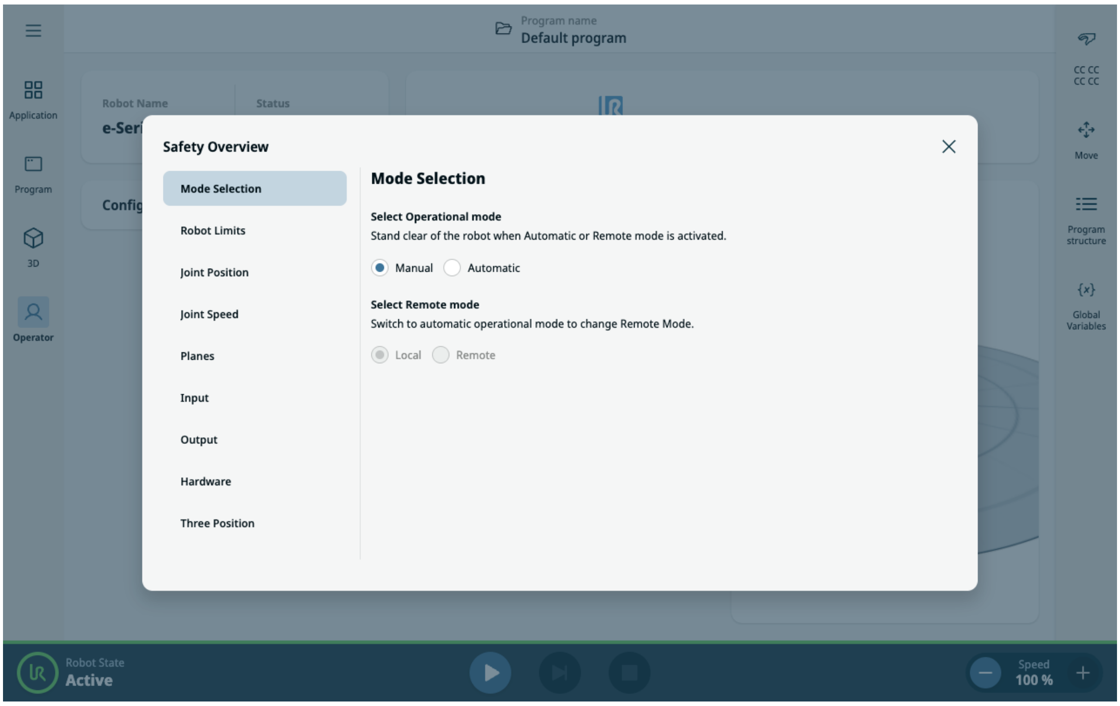
When activated, Remote Mode allows external devices to connect to key services such as the Primary Interface.

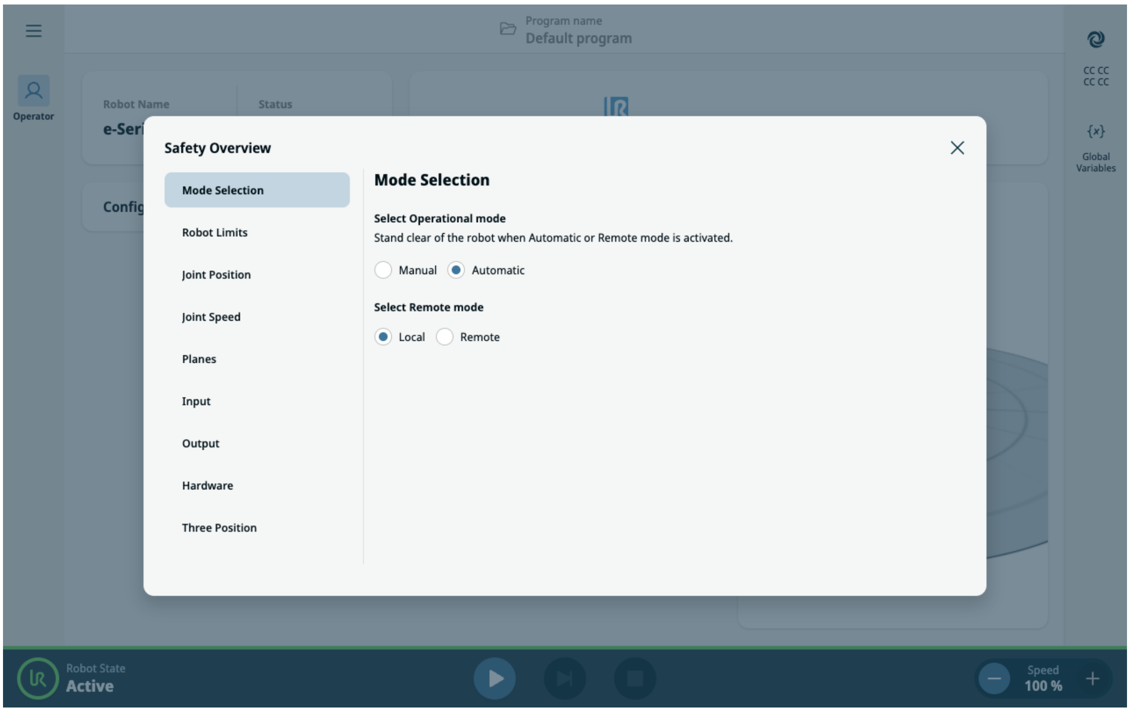
Remote Mode can be toggled via a dedicated switch in the Safety Overview dialog.

1. Go to the safety overview in the main screen.
2. Click Mode Selection.
3. You can now select Automatic and then Remote.

"Local" is toggled as a default.
"Remote" is only active when the application is in Automatic mode.

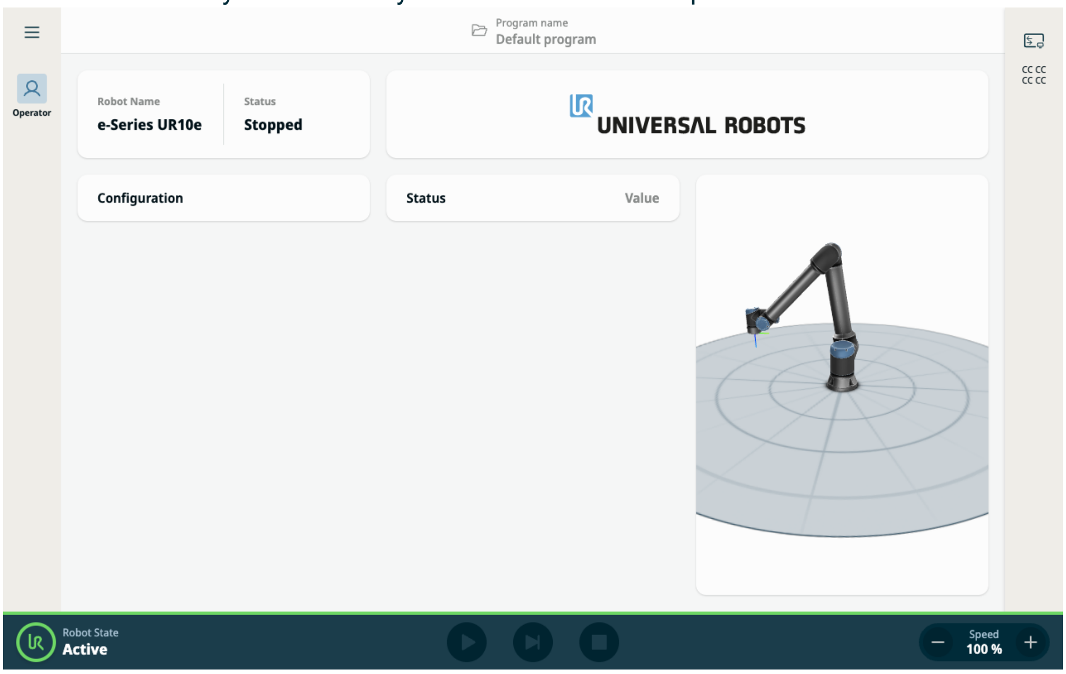
Toggle Access





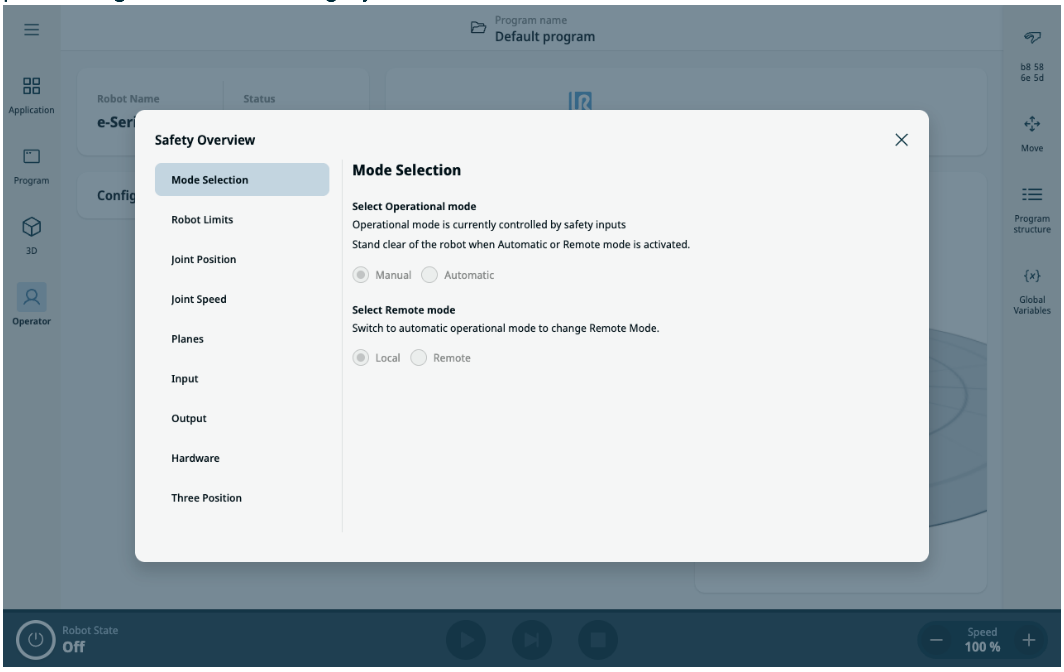
Secure Lockdown

While in Remote Mode, the PolyScope X interface enters a secure, read-only state. All editing and control actions are disabled, and only the Operator Screen remains accessible in view-only mode. Additionally, a Remote Mode icon is displayed above the Safety checksum to clearly indicate the system is under remote supervision.



I/O Controlled Safety

If the robot's operational mode is governed by an I/O signal, switching to Manual mode via I/O will automatically revert Remote Mode back to Local Mode.
This feature ensures a safe and structured environment for remote monitoring, while preserving local control integrity when needed.

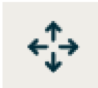


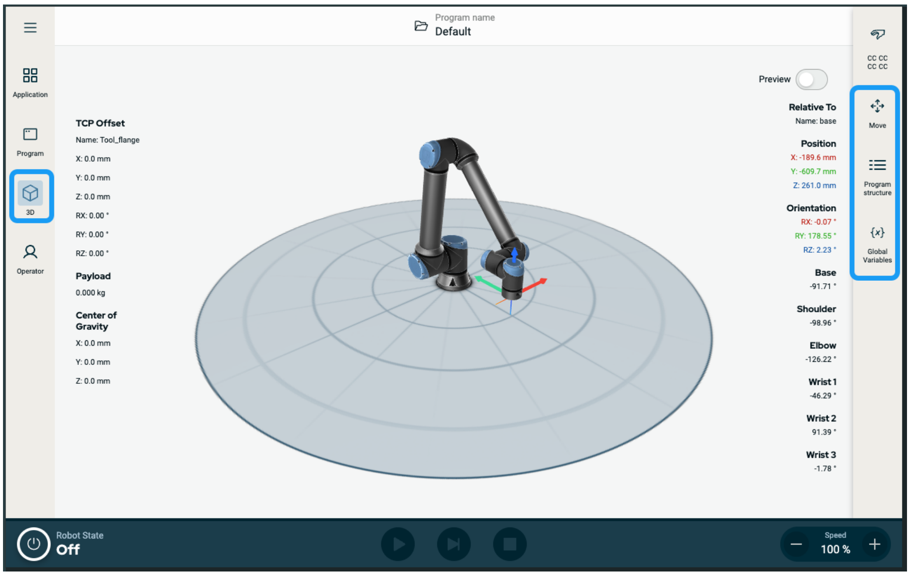
6. PolyScope X Overview

Overview

PolyScope X is the graphical user interface (GUI) installed on the teach pendant that operates the robot arm via touch screen. The PolyScope X interface allows you to create, load, and execute programs.

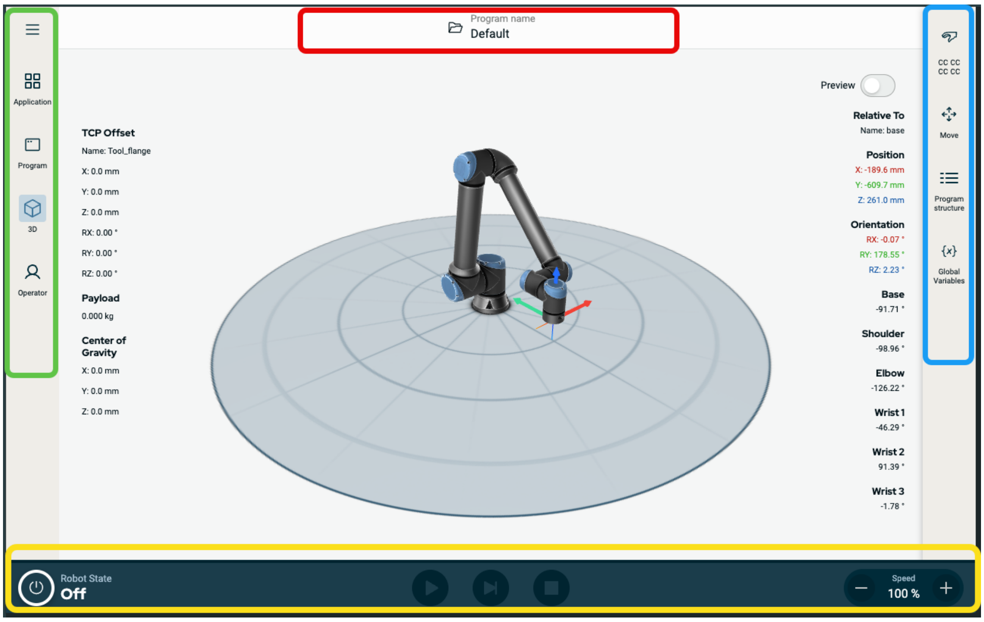
To view Main Screen

1. Tap the 3D Viewer icon  on the main navigation. This gives you a three-dimensional view of the robot arm in X-Y-Z coordinates.
2. To maximize the 3D viewing area, collapse the right drawer using the sidebar:
   - Tap once the Move icon 
   - Tap twice the Program structure icon 
   - Tap twice the Global Variables icon 



Screen Layout

The PolyScope X GUI is divided as shown in the following illustration:



- **Header** - in red-border box. Also called **system manager**.

  Contains a folder to load, create, and edit programs and access URCaps.
- **Main Navigation** - in green-border box. Also called **navigation hub**.

  Contains icon/fields to select a main screen:
  - Hamburger icon
  - Application
  - Program
  - 3D Viewer
  - Operator Screen
- **Sidebar** - in blue-border box. Also called **multitasking panel**.

  Contains icon/fields to select a multitask screen:
  - Safety checksum icon
  - Move
  - Program structure
  - Global Variables
- **Footer** - in yellow-border box. Also called **robot control bar**.

  Contains buttons to control robot state, speed, and program run/play.

Screen Combinations

The main screen and the multitask screen make up the operating screen combination for the robot.
The multitask screen is independent of the main screen so you can do separate tasks. For example, you can configure a program in the main screen while moving the robot arm in the multitask screen. You also can hide the multitask screen if it is not needed.

- **Main screen**

  Contains fields and options to manage and monitor robot actions.
- **Multitask screen**

  Contains fields and options often relating to the main screen.

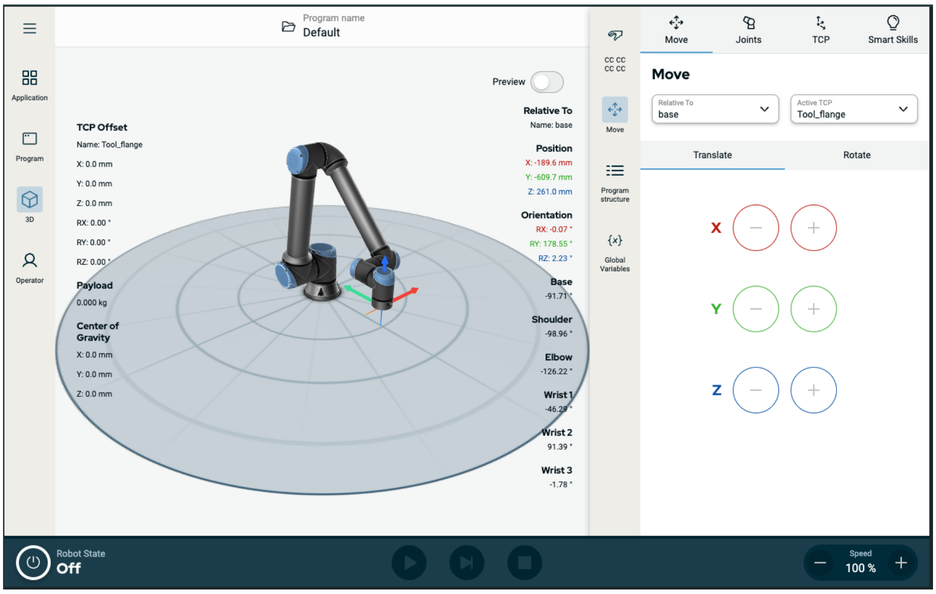


Figure 1.1: *Main screen and multitask screen*

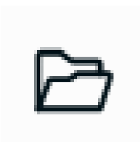
To show/hide the Multitask Screen

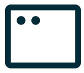
1. In the sidebar, tap any field to show the multitask screen.

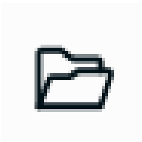
   The sidebar expands to the middle of the screen so the multitask screen becomes visible.
2. Tap the currently selected field in the sidebar to hide the multitask screen.

6.1. Icons

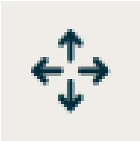
Header Icons

Icon	Title	Description
	Program name	Gives access to System Manager. Allows you to load, save, and add programs and URCaps files.

Main Navigation Icons	Icon	Title	Description
		More	Access information about robot version, serial number, and settings.
		Application	Configures and sets up the robot arm settings and safety, including end effectors and communication.
		Program	View and modify robot programs.
		3D	Enables control and regulation of robot movement in X, Y, Z coordinates.
		Operator	Operates the robot using prewritten programs and shows the status of the robot.

Icons Inside the Hamburger Icon	Icon	Title	Description
		System Manager	Gives access to System Manager. Allows you to load, save, and add programs and URCaps files.
		About	Displays information about robot version and serial number.
		Settings	Configures system settings, such as language, units, password, and security.
		Reload	A safe function to apply the default settings defined in the application.
		Shutdown	To restart, power on and off the robot.

Sidebar Icons

Icon	Title	Description
CC CC CC CC	Safety Checksum	Displays the active safety checksum and gives access to detailed parameters of each robot arm parts, and changes the operational mode.
	Move	Comprehensive function for robot movement, detailing the joints, TCP, flange, base.
	Program structure	Provides structural overview of the main program, modules and functions. Access to add modules.
{x}	Global Variables	Provides access to global variable names and their values.

Footer Icons	Icon	Title	Description
		Initialize	Manages the robot state. When RED, press it to make the robot operational. • Black, Power off. The robot arm is in a stopped state. • Orange, Idle. The robot arm is on, but not ready for normal operation. • Orange, Locked. The robot arm is locked. • Green, Normal. The robot arm is on and ready for normal operation. • Red, Error. The robot is in a fault state, such as e-stop. • Blue, Transition. The robot is changing state, such as brake releasing.
		Play	Starts the current loaded program.
		Step	Allows a program to be run single-stepped.
		Stop	Halts the current loaded program.
	Speed 100 %	Speed Slider	Shows in real time the relative speed at which the robot arm moves, taking safety settings into account.
	Max 250 mm/s	High Speed Manual Mode	The High Speed Manual slider is only available in manual mode when a Three-Position Enabling Device is configured. High Speed Manual Mode allows tool speed and elbow speed to temporarily exceed the default speed limit.

Main Screen Icons

Icon	Title	Description
	Move up	To move up a command node in a program tree.
	Move down	To move down a command node in a program tree.
	Revert	To revert a recent move of a command node in a program tree.
	Undo revert	To undo revert a recent move of a command node in a program tree.
	Suppress/ Unsuppress	To suppress and unsuppress a command node in a program tree.
	Copy	To copy a command node to another program tree.
	Paste	To paste a command node to another program tree.
	Cut	To cut a command node from a program tree.
	Delete	To delete a command node in a program tree.

6.2. Touch Screen

Description

The **Teach Pendant touch screen** is optimized for use in industrial environments. Unlike consumer electronics, teach pendant touch screen sensitivity is, by design, more resistant to environmental factors such as:

- Water droplets and/or machine coolant droplets
- Radio wave emissions
- Other conducted noise from the operating environment

Using the Touch Screen

The touch sensitivity is designed to avoid false selections on PolyScope X and to prevent unexpected motion of the robot.
For best results, use the tip of your finger to make a selection on the screen. In this manual/handbook, this is referred to as a **tap**.
A commercially available stylus may be used to make selections on the screen, if desired.
The preceding section lists and defines the icons/tabs and buttons in the PolyScope X interface.

6.3. Header

Description

The header solely contains the **Program name** menu.

To open Header

When you tap the **Program name** menu, the **System Manager** screen appears, which enables you to do several things:

- View a list of all installed programs
- See the loaded program, marked **Active** in green
- Create new programs
- Open a created program
- Import and export a program
- Load a program

- Install URCaps
- Ensure the software license
- Add a script file
- Write a program description
- See program details
- View script

6.4. Main Navigation

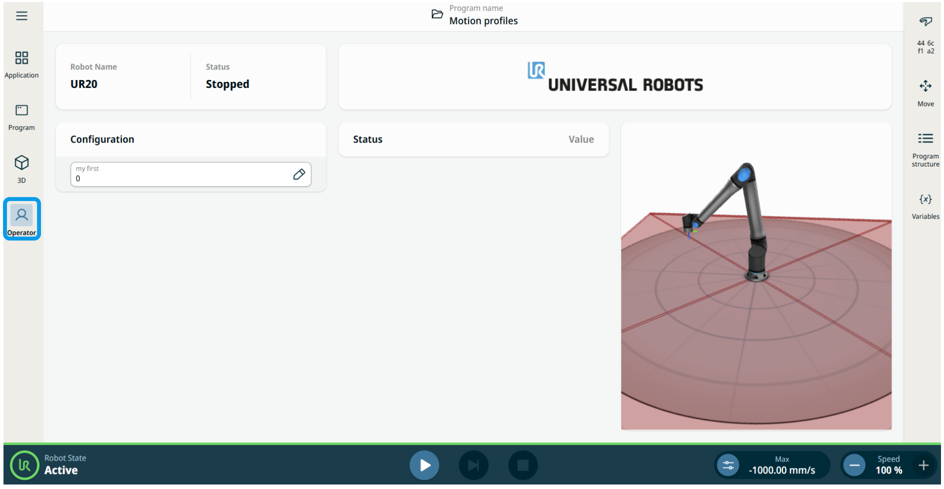
Description

The main navigation, also called the navigation hub, contains five settings/menus:

- Hamburger menu icon
- Application
- Program
- 3D Viewer
- Operator Screen

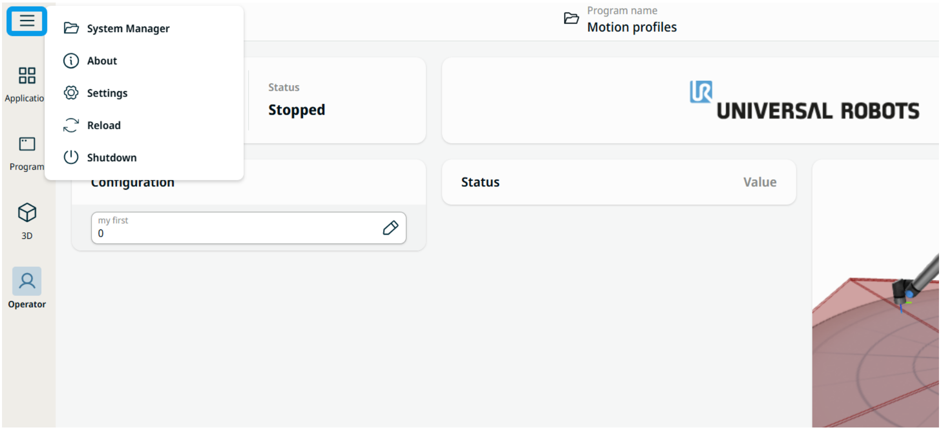
More about the Main Navigation

When you open PolyScope X, your main screen is by default the Operator Screen. The Operator icon, highlighted in blue, indicates this.

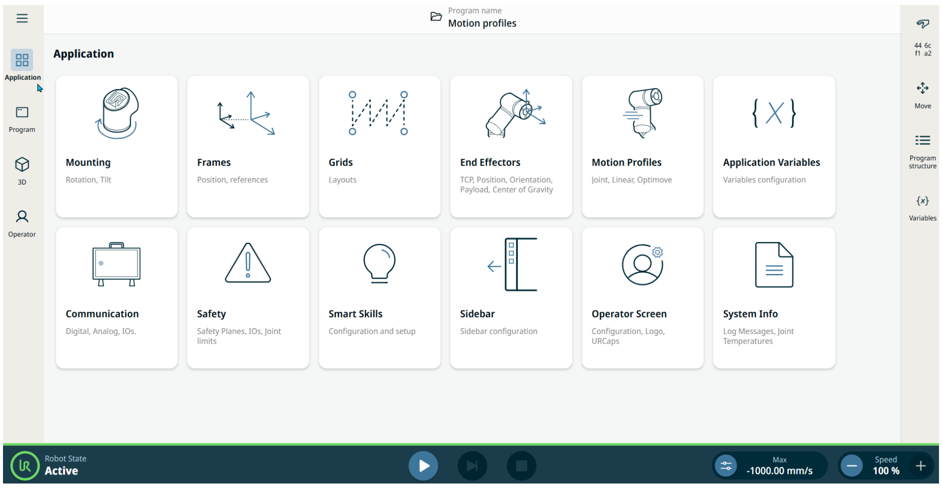


When you tap an icon, it is highlighted blue in the main screen, which indicates it is active. The hamburger icon is an exception to this.

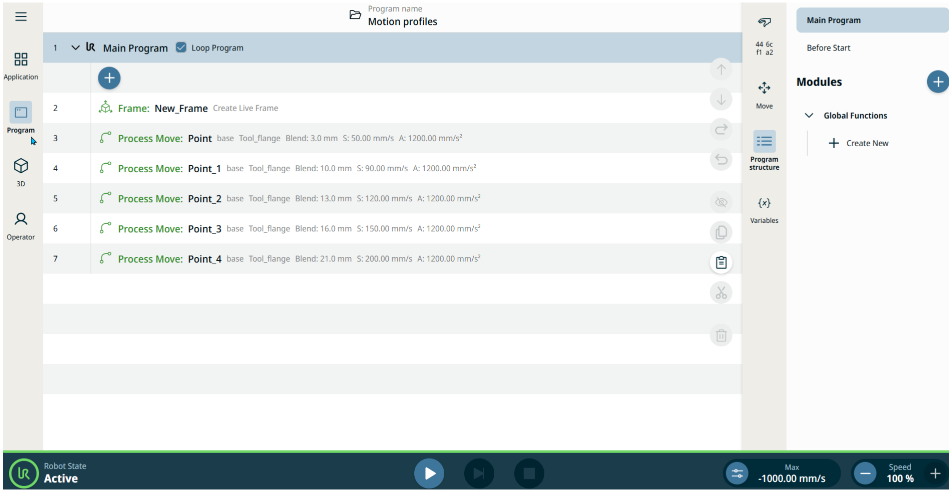
Tap hamburger icon



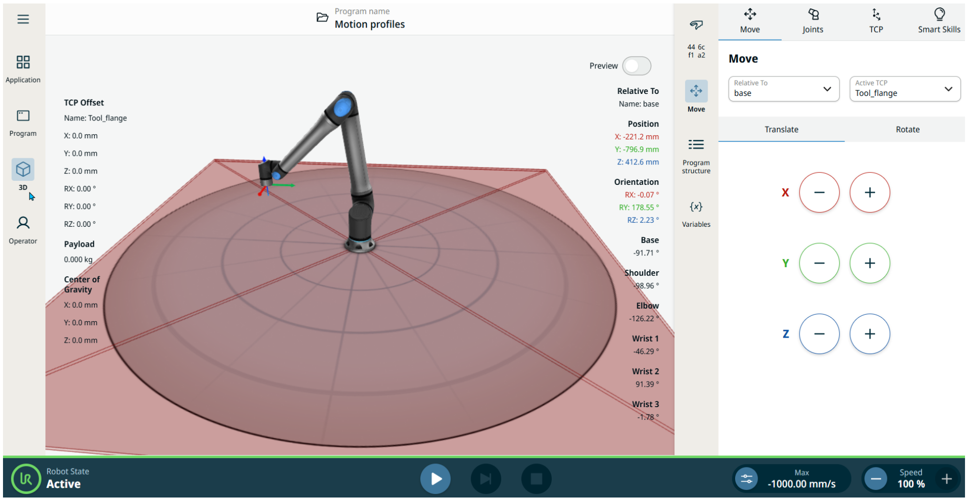
Tap Application



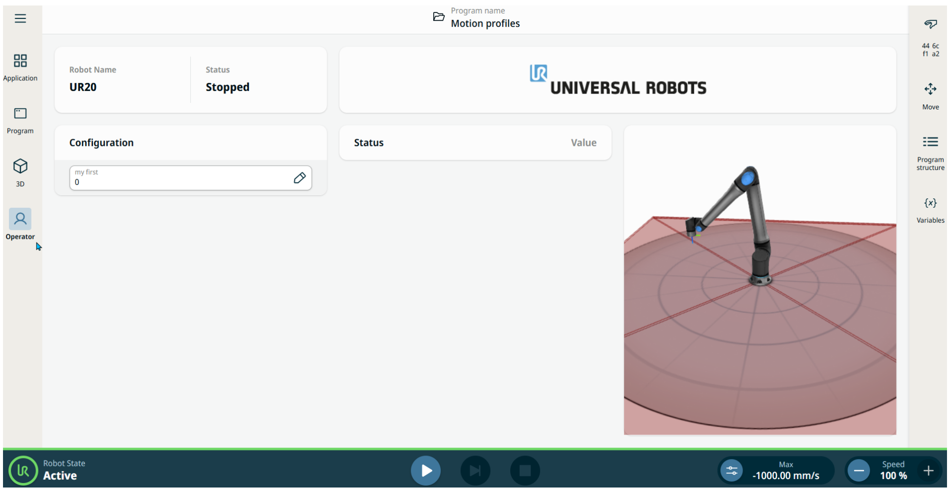
Tap Program



Tap 3D



Tap Operator

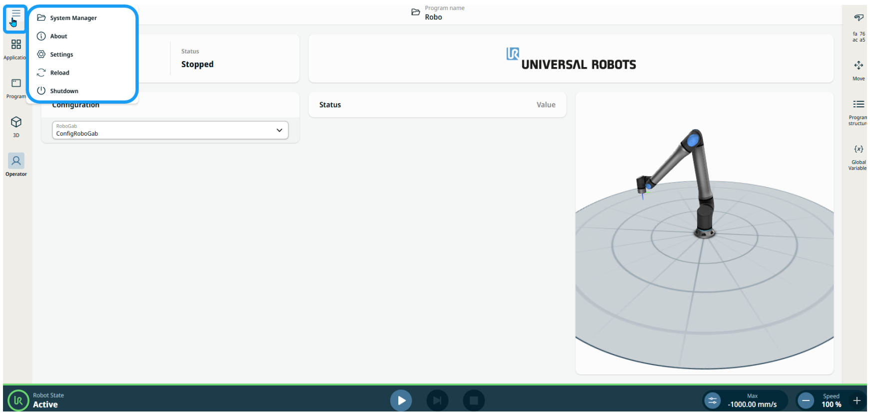


6.4.1. Functions Inside the Hamburger Icon

Description

The **hamburger icon** contains five functions of PolyScope X:

- System Manager
- About
- Settings
- Reload
- Shutdown

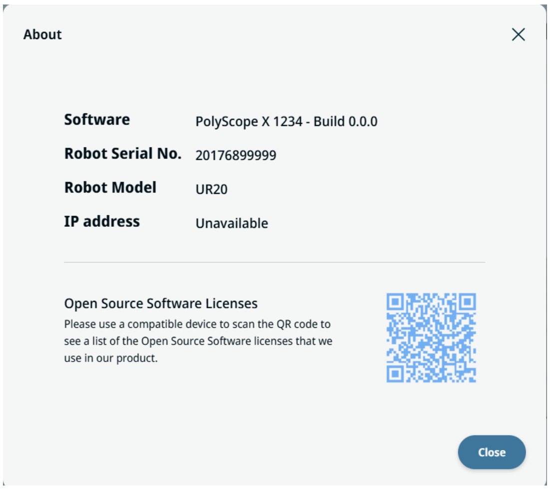


System Manager

The **System Manager** has the same function as with the header. See Header.

About

The **About** function requires no further action to perform in another field or tab. When you tap About, you see the software version, robot serial number, and the QR code of the Open Source Software licenses. The **robot model** and its **IP address** are shown to provide users with quick access to essential system information.



Settings

The **Settings** function gives you access to **General**, **Password**, **Connection**, and **Security** settings. Except for System in the general settings, all are password protected.

General.
This setting contains System and Update.

System setting enables you to change the language, theme, and units used in the system.
Update setting is where you are asked to insert a USB storage and check for update.

Password
This setting contains Operational Mode, Safety, and Admin.

Operational Mode password is used when switching between automatic and manual mode.
Safety password is used to change safety settings.
Admin password is used to change system settings. Observe extra care when changing this, as admin password is nonrecoverable. When password is not retrieved, software reinstallation is needed.

Connection
This setting contains Network and UR Connect.

Network enables you to apply network configuration after choosing either DHCP or Static.
UR Connect enables you to connect to myUR Cloud, which allows you to access your robot data anywhere.

Security
This setting contains Secure shell, Permissions, and Services.

Secure shell manages SSH access to the system.
Permissions enables you to select system pages to be protected by Admin password.
Services lets you disable or enable interfaces.

Reload

The **Reload** function requires no further action to perform in another field or tab. When you tap Reload, the default settings defined in the application will be applied.

Shutdown

The **Shutdown** function requires no further action to perform in another field or tab. When you tap Shutdown, you are given the choice to restart or power off. Program state will be saved before restarting or powering off the robot.

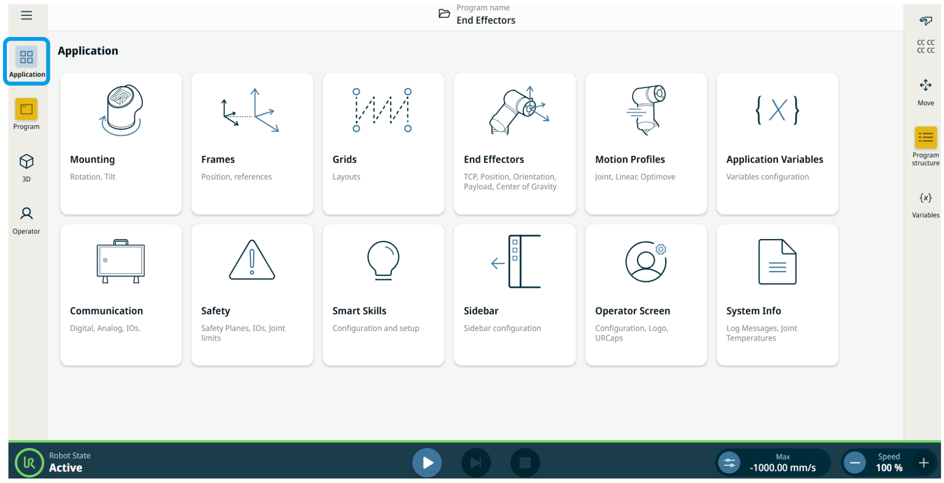
6.4.2. Application Tab

Description

The **Application** tab allows you to configure the settings which affect the overall performance of the robot and PolyScope X.

To access Application

Tap **Application**.



The **Application screen** appears, which shows twelve application submenus to use for robot movement, configuration, and information.

- Mounting
- Frames
- Grids
- End Effectors
- Application Variables
- Motion Profiles
- Communication
- Safety
- Smart Skills
- Sidebar
- Operator Screen
- System Info

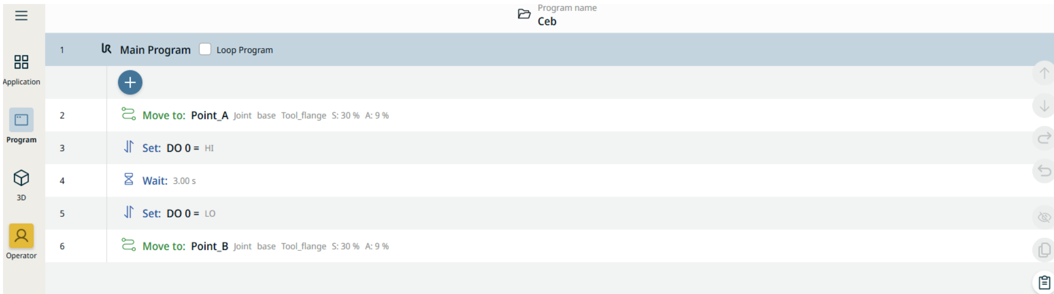
6.4.3. Program Menu

Description

The **Program** menu gives you access to the commands and clipboard toolboxes, which contain the basic command nodes.

To access Program

When you tap the Program icon, the Main Program screen becomes the main screen. The active program is seen in it, along with the details of the command nodes used in the program tree.
Tap the add icon to access the Commands and Clipboard toolboxes. See 16.3 Command Nodes on page 117.



6.4.4. 3D Viewer

Description

The **3D** viewer shows in the main screen a three-dimensional view of the robot arm in X, Y, Z coordinates. Measurements of the robot arm parts position, orientation, payload, center of gravity, and TCP offset are given in both sides of the main screen.
You can see the multitask screen, which enables you to move the robot using the editable fields, tabs, and plus-minus button.

To view in 3D

Tap the 3D icon, and the main screen and multitask screen appear simultaneously.
Use the following touch gestures applicable in the **main screen**:

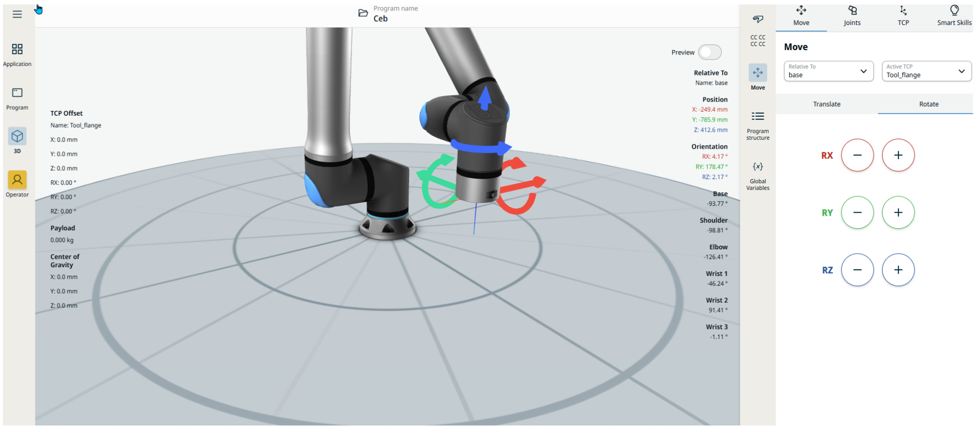
- **Press and rotate** clockwise or counterclockwise.

  Function: Changes the viewing direction.
- **Pinch**. Touch the screen with two fingers and bring them closer together.

  Function: Zooms out the robot image.
- **Spread**. Touch the screen with two fingers and move them apart.

  Function: Zooms in the robot image.

In the **multitask screen**, tap the plus-minus button of X, Y, Z to translate and the RX, RY, RZ to rotate. A responsive, coordinated movement of the robot arm in the main screen is shown simultaneously.
See the color-coded indications of the straight and circular arrow in the robot arm.



6.4.5. Operator Screen

Description

The **Operator screen** shows information of the robot name, robot status, configuration, status, and a smaller 3D view of the robot arm.
The operator screen gives users the following benefits:

- Make changes without support or the least complex support in multiple workcells.
- Avoid accidental program modification.
- Manage one flexible program instead of multiple separate programs across different processes.

6.5. Sidebar

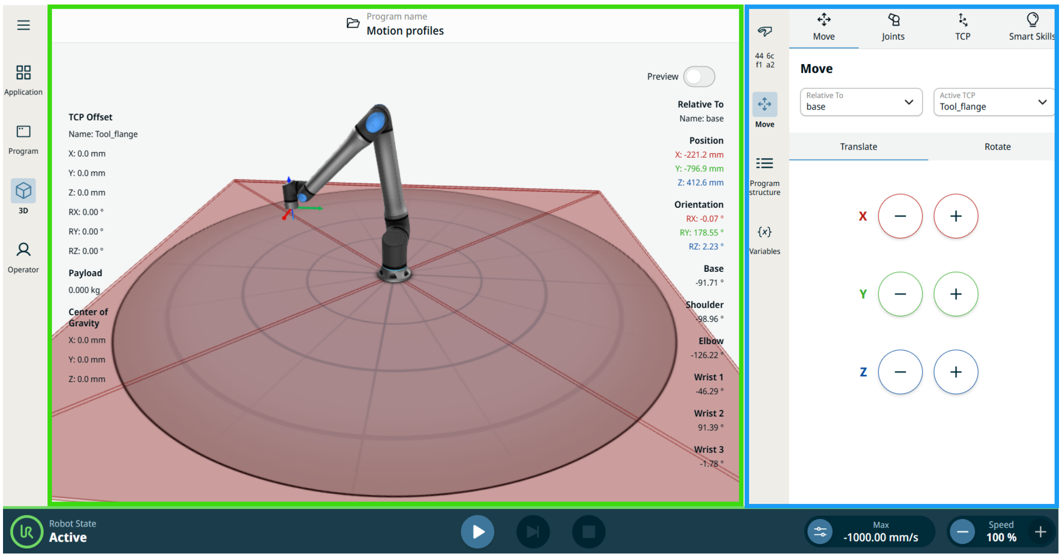
Description

The sidebar contains four menus:

- Safety Checksum
- Move
- Program Structure
- Global Variables

Using the Sidebar

When you tap the menus in the sidebar, their part of the screen will expand to the center. That screen is called the multitask screen (inside the blue box).
Any changes and adjustments done inside the multitask screen have the robot arm's graphical responses in the main screen (inside the green box).



6.5.1. Safety Checksum

Description The Safety Checksum icon displays your applied robot safety configuration. The checksum changes if and only if the safety configuration is changed.
The icon has three states/appearances:

- A **hand** 

indicates that the control station is in control and the operational mode is set to manual.
- A **whirling process**  indicates that the control station is in control and that the operational mode is set to automatic.
- A **remotely connected system**  indicates that control is external to the station.

Safety Checksum is the only menu with non-labeled icon 

.
The Safety Overview screen is divided in two:

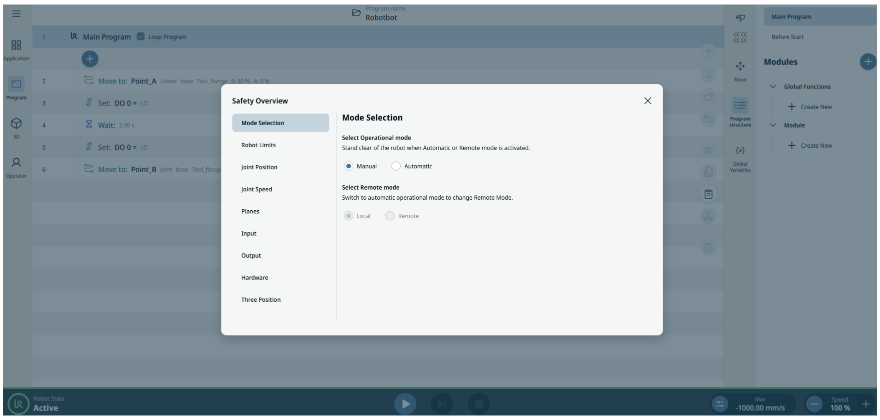
- Left side is smaller and contains the following safety item settings:

Mode Selection	Robot Limits	Joint Position	Joint Speed
Planes	Tool Position	Tool Direction	I/O
Hardware	Safe Home	Three Position	

- Right side contains a tabular data of each safety item setting.

To access Safety Overview

1. Tap the **Safety Checksum** icon.
2. Tap **Mode Selection** on the left side.
3. In **Select Operational mode**, tick either **Manual** or **Automatic**.



6.5.2. Move Menu

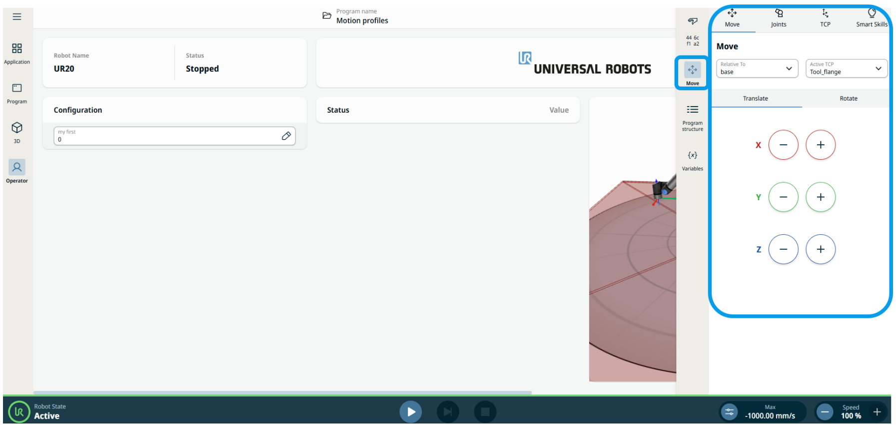
Description

The **Move** menu provides the functions to move the robot arm directly, either by translating and/or rotating the robot tool or by moving robot joints individually.

Using the Move menu

1. Tap the **Move icon** on the sidebar.

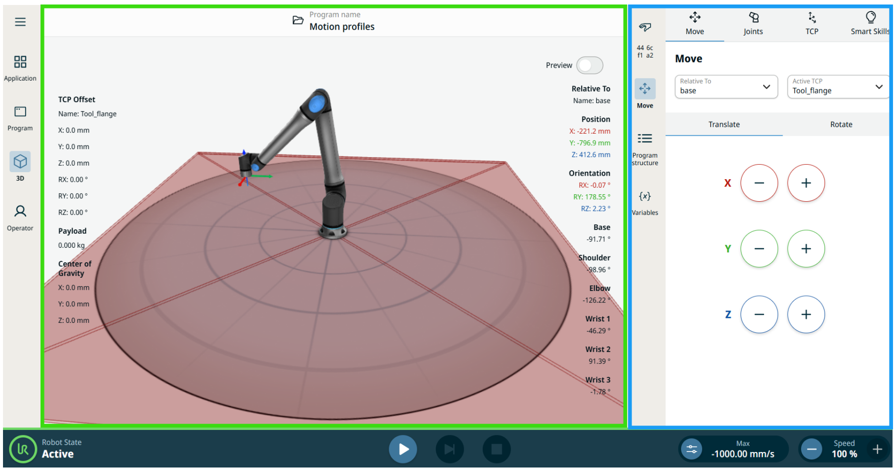
   The multitask screen expands to the center. You can see the fields and tabs related to the move menu.



2. Tap the **3D icon** on the main navigation.

   The main screen shows the robot arm in X, Y, Z coordinates with the measurement details of each robot arm parts, position, and orientation.

   In the main screen, you can see the corresponding movement(s) of the robot with what you input in the multitask screen.

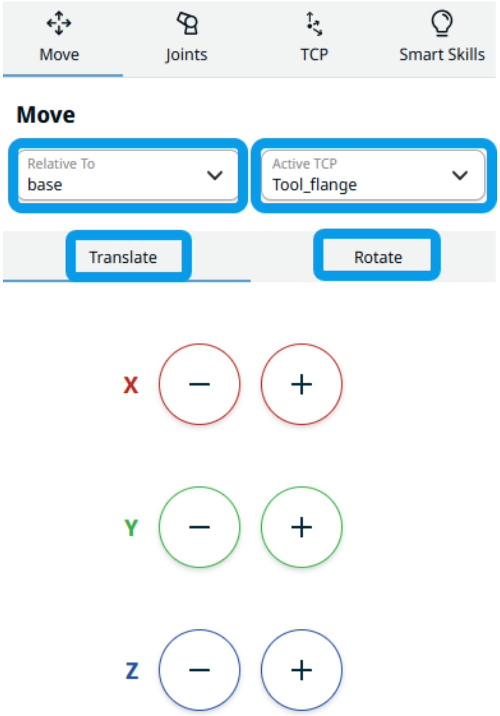


Move Tool

Using the Move tool

The Move tool contains two fields and two tabs:

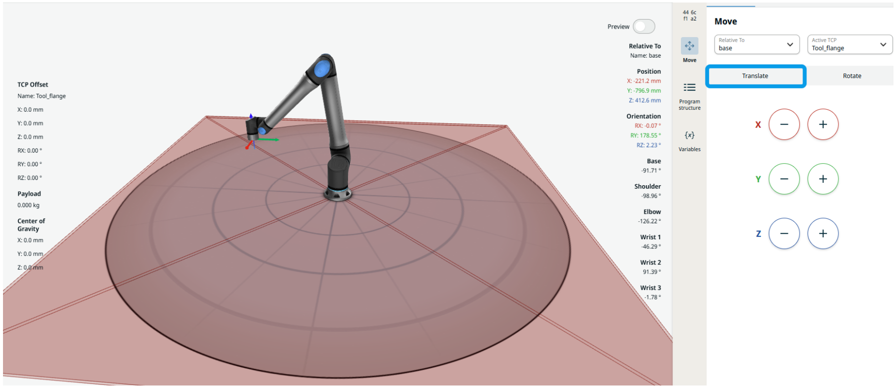
- Relative To
- Active TCP
- Translate
- Rotate



1. Tap the **Relative to** field.
   Four choices are given:
   - world
   - base
   - flange
   - tcp
2. Select the relative position you want the robot to move.
3. Tap the **Active TCP** field and select your preference.

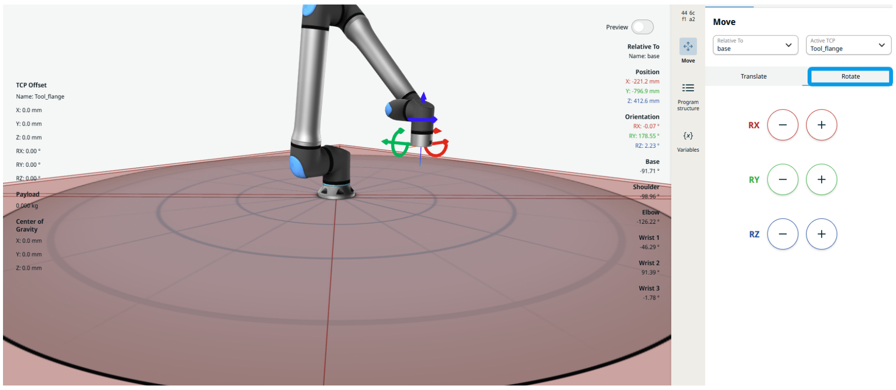
4. Tap the **Translate** tab.
5. Tap the **plus-minus** button of X, Y, Z coordinates to move the robot arm in an indicated direction.

   In the main screen, you can see the corresponding movement, as indicated by the red, green, and blue straight arrows of the robot arm.



6. Tap the **Rotate** tab.
7. Tap the **plus-minus button** of RX, RY, RZ coordinates to change the orientation of the robot in the indicated direction.

   In the main screen, you can see the corresponding movement, as indicated by the red, green, and blue circular arrows of the robot arm.



Joints Tool

Using the Joints tool

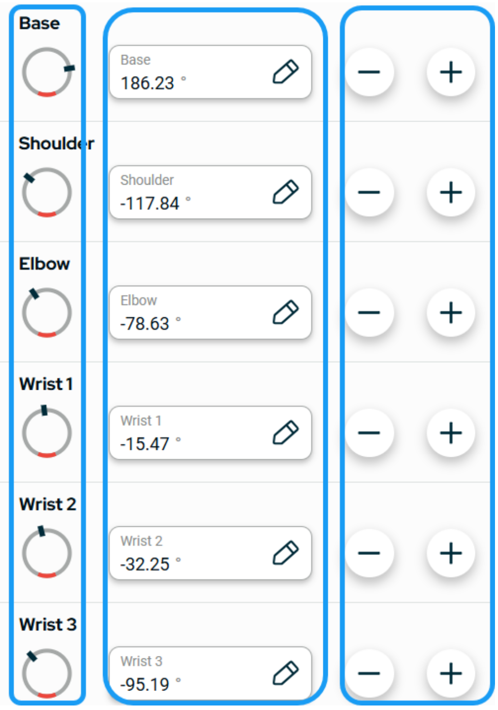
1. Tap **Joints**.

The Joints screen appears, which contains the six parts of the robot arm:

- Base
- Shoulder
- Elbow
- Wrist 1
- Wrist 2
- Wrist 3

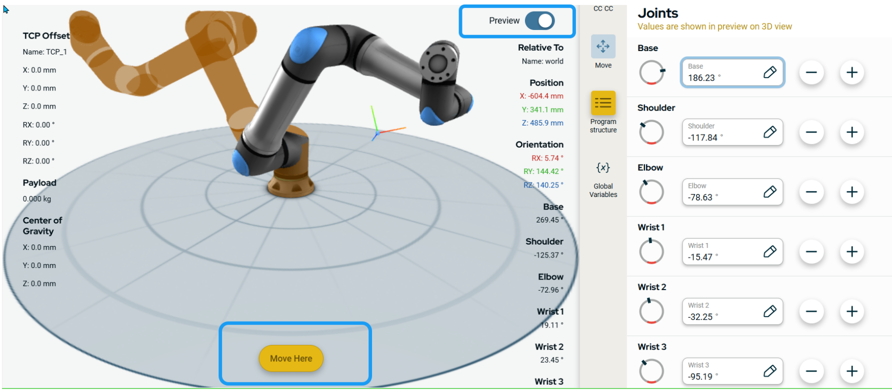
Each robot part gives you three adjustment tools so the position measurement be accurately achieved:

- Rotating ring
- Editable field
- Plus-minus button



2. Rotate the ring clockwise or counterclockwise to reach your preferred value/degree.
3. Tap the edit field and enter the value.
4. Tap **Confirm**.

   Note: The rotating ring and the edit field have similar input for preview of the robot arm.

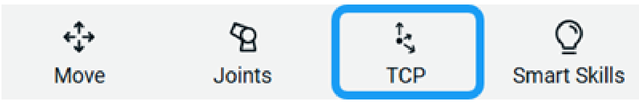
   A preview of the robot arm position/orientation appears in brown color. You can disable the preview when you tap the **Preview button** to the left.

5. When you want to preview the whole robot arm movement, tap the brown **Move Here** button below the X, Y, Z coordinate space of the robot arm.
6. Tap the **plus-minus** button to adjust and reach your preferred measurement of the robot arm joints.

TCP Tool

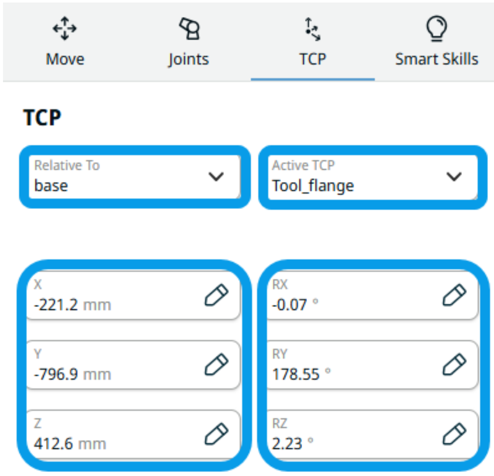
Using the TCP tool

1. Tap **TCP**.



The TCP screen appears which contains:

- Relative to field
- Active TCP field
- X tab
- Y tab
- Z tab
- RX tab
- RY tab
- RZ tab



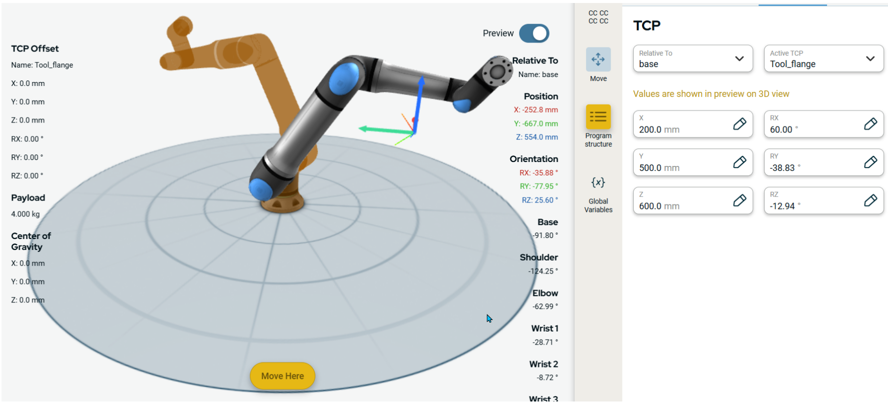
2. Tap the **Relative to** field.

   Four choices are given:

   - world
   - base
   - flange
   - tcp

3. Select the relative position you want the robot to move.
4. Tap the **Active TCP** field and select your preference.
5. Tap the **X, Y, Z** tabs and **RX, RY, RZ** tabs and edit the value for each.

   A preview of the robot's position is shown in the robot arm with brown color.

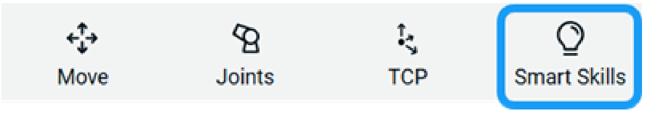


6. To preview the whole robot arm movement, tap the brown **Move Here** button below the X, Y, Z coordinate space of the robot arm.

Smart Skills Tool

Using the Smart Skills tool

1. Tap **Smart Skills**.



The **Smart Skills** screen appears which contains two fields, six large icons, and a button.

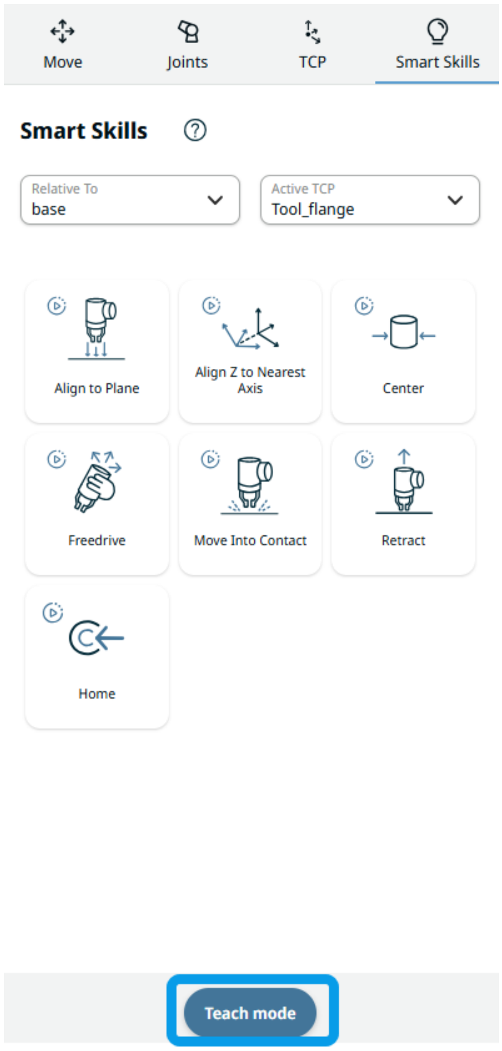
The two fields are:

- Relative to
- Active TCP

The seven Smart Skills icons are:

- Align to Plane
- Align Z to Nearest Axis
- Center
- Freedrive
- Move into Contact
- Retract
- Home

The **Teach mode** button is at the bottom.



2. Tap the **Relative to** field.

   Four choices are given:

   - world
   - base
   - flange
   - tcp

3. Select the relative position you want the robot to move.
4. Tap the **Active TCP** field and select your preference.
5. Select and tap the smart skill icon of your preference.

   In the main screen, you can see the corresponding movement of the robot arm.

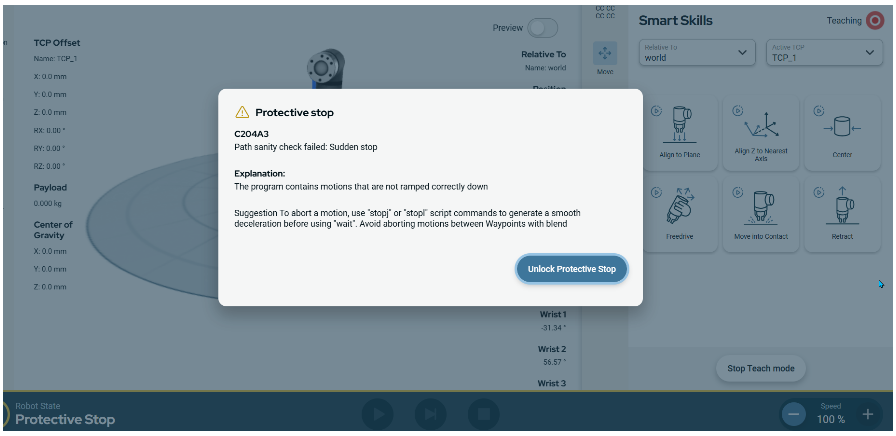
6. Tap the **Teach mode** button to program the robot by demonstration.

   You see a red blinking light on the upper right of the multitask screen, signifying teaching is ongoing.

7. Apply a **Smart Skill** and wait for it to stop, or stop it yourself.

   A program node is generated in the program tree.

8. If a **Protective stop** caution appears in the main screen, do the necessary commands to properly and safely program the robot.



6.5.3. Program Structure

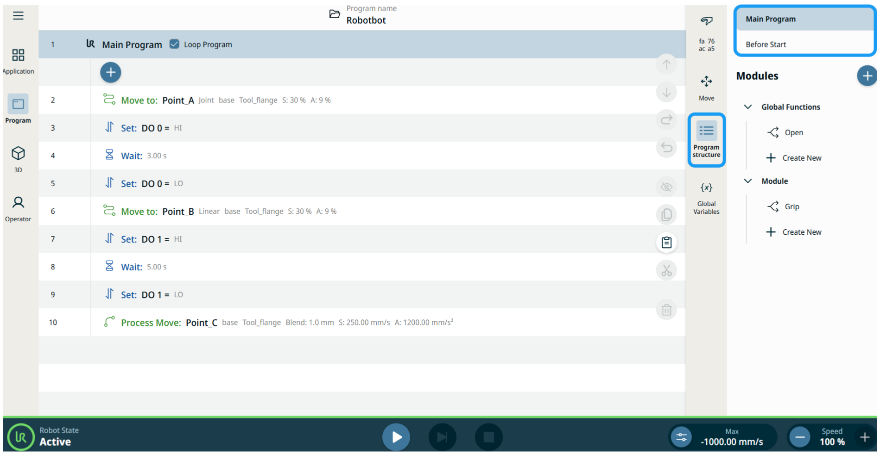
Description

The **Program Structure** menu lists the order and functions to create robot programs.

To access Program Structure

Tap the **Program Structure** menu.
Access to two tabs related to the program menu is provided:

- Main Program
- Before Start



Tap either Main Program or Before Start, and you can see the corresponding responses in the main screen.
Select **Before Start** to add instructions or program nodes that are executed before the main program starts. Variables created in creating program are always done under **Before Start**.

6.5.4. Global Variables

Description

Global Variables contain the list of all variables in the robot program with global scope. These encompass ones created and found in:

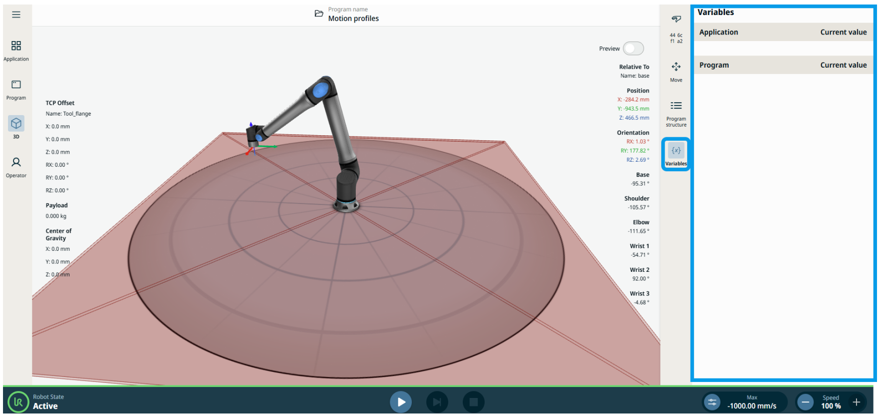
- Assignment nodes
- Operator Screen configuration
- Status
- Frames
- Waypoints

If the values of these variables change while the program is running, their live values is shown.

To access Global Variables

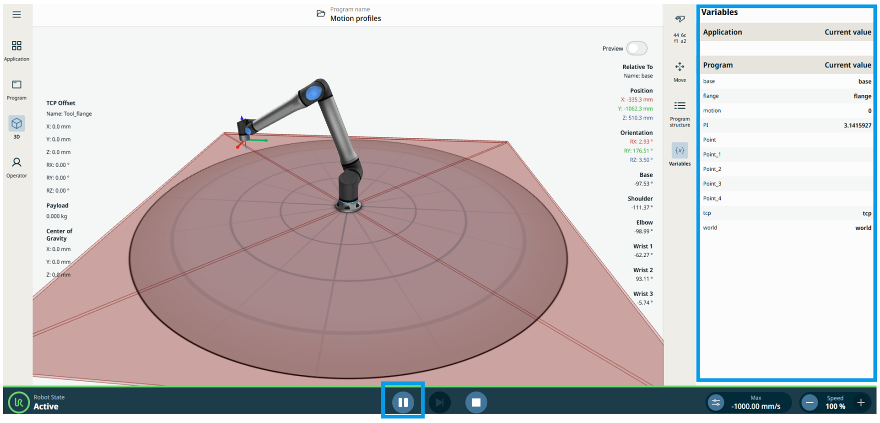
1. You can choose to view the main screen in **3D** view or **Operator** screen.
2. Tap **Global Variables** in the sidebar.

   You see that the **Name** and **Value** columns are blank.



3. Tap the **Play** button in the footer and view the variable names and values.

   The number of names and values depend on the program you created and how many commands you used.



6.6. Footer

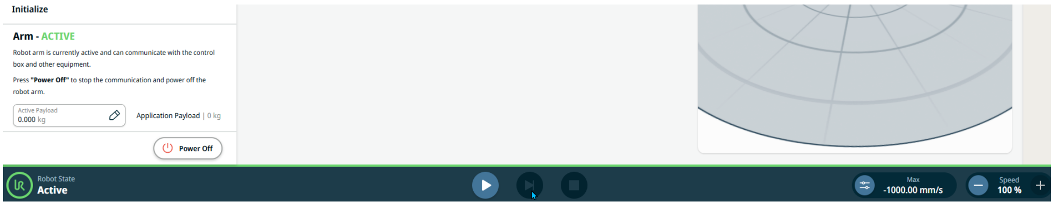
Description

The **Footer**, also called robot control bar, contains buttons to control robot state, program play and stop, and control sliders for manual high speed and robot speed.

Functions in Footer

In the footer, you initialize the robot arm:

- Power On
- Power Off
- Edit the active payload
- Unlock
- Enable manual high speed



7. Settings

Description

The settings in PolyScope X can be accessed via the hamburger menu in the main navigation.
You can access the following sections:

- General
- Password
- Connection
- Security

General Settings

In the general settings, you can change the preferred language, units of measurements, etc. You also update the software from the general settings.

Password Settings

In the password settings, you can find the default passwords, and how to change them to the preferred and secure passwords.

Connection Settings

In the connection settings, you can set network settings such as IP address, DNS server, etc. Settings related to UR Connect is also found here.

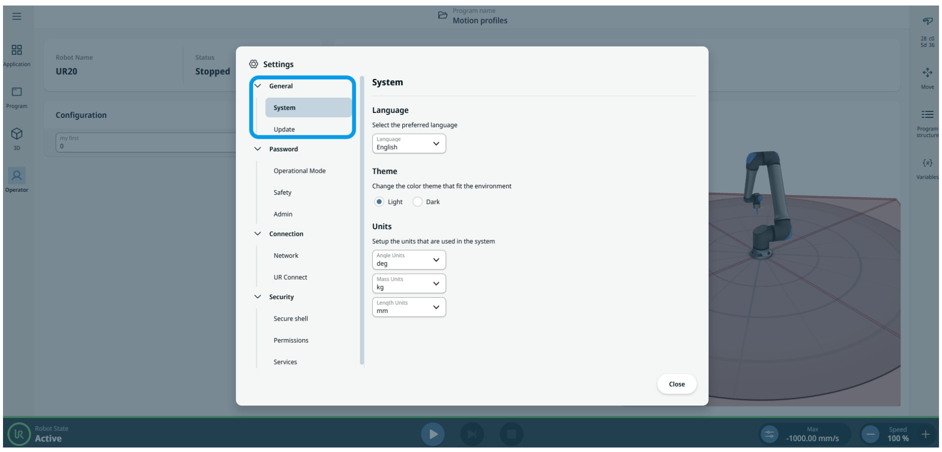
Security Settings

The security settings related to SSH, admin password permissions and enabling/disabling of various services in the software.

7.1. General Settings

Description

The **General** setting of PolyScope X provides you the function of System and Update.



To open General setting

1. Tap the hamburger menu on the main navigation.
2. Tap **Settings** tab.

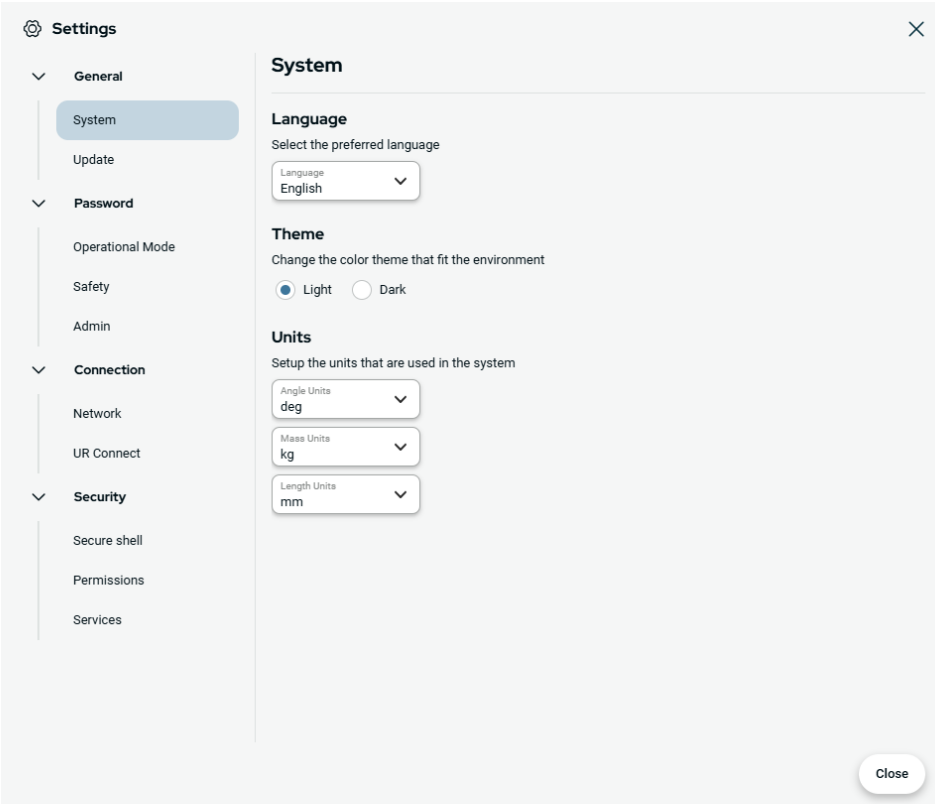
   A left sidebar opens that contains the **General** tab.
3. Under the General tab, tap **System** bar.

   You now have access to **Language**, **Theme**, and **Units** menu to suit your preference.
4. Under the General tab, tap **Update** to see your software version and check for updates.

7.1.1. System

Description

The system settings contain underlying settings that is set once and very rarely changed.



Languages

You can change the preferred language for the UI.
Hit the language selector and find your language.

Theme

You can change the preferred language for the UI.
Hit the language selector and find your language.

Units

It is possible to change the precision of the software in the settings. Please make sure that you verify this setting, before you start programming the robot program.

Angle Units:
You can change between degrees and radian.

Mass Units:
You can change between kilogram and gram.

Length Units:
You can change between meter and millimeter.

7.1.2. Update

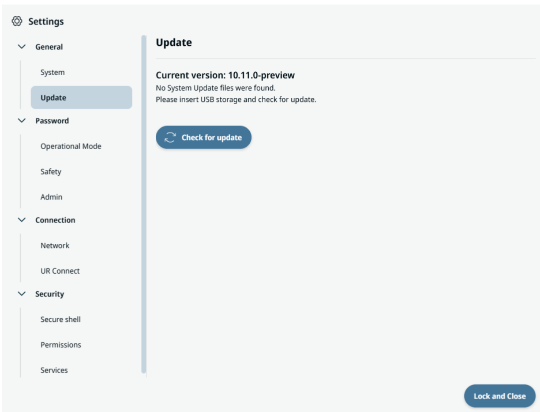
Description

When you receive your robot, it is recommended that you update the software to the most recent release from Universal Robots.
Please use this link to find the latest software update: Download Latest Software

Update SW

This is how you update the software:

1. Tap the hamburger menu in the main navigation.
2. Go to Update in the general settings.
3. Insert the USB with the software update.
4. Tap the button "Check for update".



7.2. Password

Description

In the password settings in PolyScope X, you can find three different types of password.

- Operational Mode
- Safety
- Admin

It is possible to set the same password in all three instances, but it is also possible to set three different password to separate access and options.

7.2.1. Password - Admin

Description

All options under Security are protected by an Admin password. The Admin password protected screens are locked by a transparent overlay rendering the settings unavailable. Accessing the Security allows you to configure the settings in the following:

- Secure Shell
- Permissions
- Services

The settings can only the modified by designated administrator/s.
Unlocking any one of the options under Security, also unlocks the other options until you exit the Settings menu.

Default Password

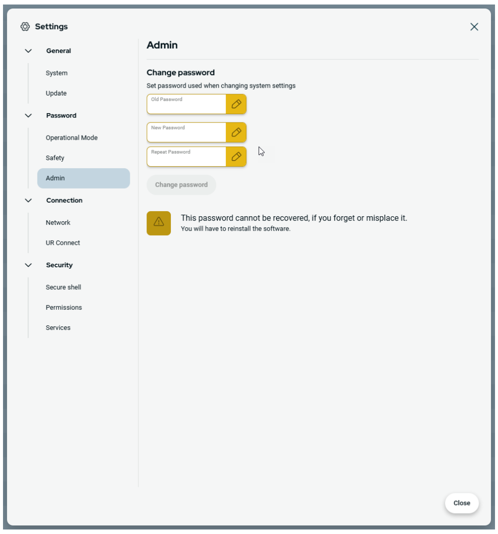
The default password for the admin password is: easybot

NOTICE
If you forget your admin password, it cannot be replaced or recovered.
You will have to reinstall the software.

To set the Admin password

Before you can use the Admin password to unlock protected screens, you have to change the default password.

1. Access the Hamburger menu and select **Settings**.
2. Under Password, tap **Admin**.
3. Change the current Admin password to a new one.
    - If this is the first time, change the default Admin password from "easybot" to a new password. The new password must be at least 8 characters long.
4. Use the new password to unlock the Settings menu and access the options under Security.



To exit the Settings menu

When one of the Security options is unlocked, the Close button in the bottom right of the Settings menu changes. The Close button is replaced by the Lock and Close button indicating security is unlocked.

1. On the Settings menu locate and tap the **Lock and Close** button.

7.2.2. Password – Operational Mode

Default Password

The default password for operational mode: operator

NOTICE

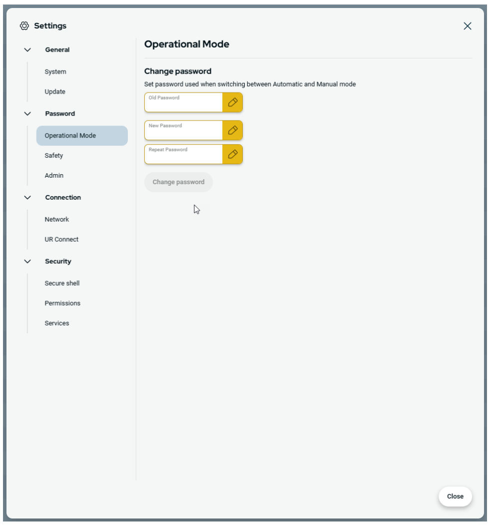
If you forget your password, it cannot be replaced or recovered.
You will have to reinstall the software.

You must use the default password, when you change the password for the first time.

Change Operational Mode password

This is how you change the password for operational mode in the PolyScope X setting.

1. Tap the hamburger menu in the main navigation.
2. Tap Settings.
3. Tap Operational Mode in the Password section.
4. Add the default password, if it is the first time.
5. Add your preferred password, at least 8 characters.



7.2.3. Password - Safety

Default Password

The default password for safety: **ursafe**

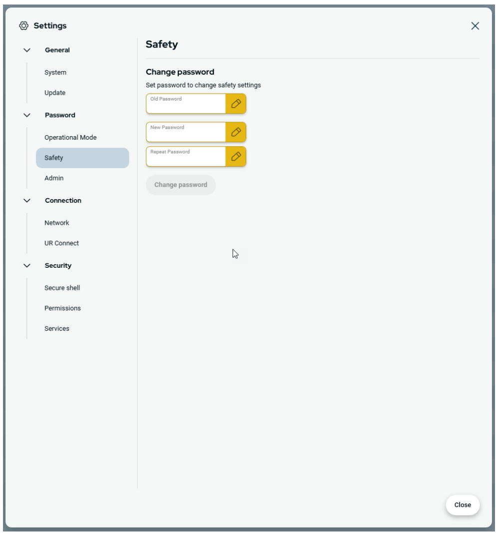
> **NOTICE**
> If you forget your password, it cannot be replaced or recovered.
> You will have to reinstall the software.

You have to use the default password, when you change the password for the first time.

Change Safety Password

This is how you change the safety password in the PolyScope X setting.

1. Tap the hamburger menu in the main navigation.
2. Tap Settings.
3. Tap Safety in the Password section.
4. Add the default password, if it is the first time.
5. Add your preferred password, at least 8 characters.



7.3. Connection

Description

The **Connection** setting of PolyScope X software enables you to access robot data anytime, anywhere.
You can access the Connection setting via the hamburger menu.

To access Connection tab

The main screen can be seen below.

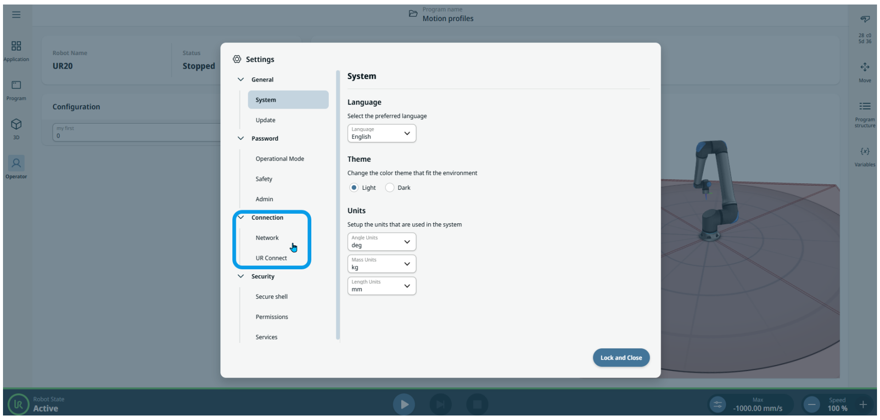
1. On the left header, tap the hamburger menu .

   Five tabs appear. **Settings** tab is the third.

2. Tap **Settings**.

- A sidebar opens with the **Connection** tab, below General and Password tabs.
- You can see **Network** and **UR Connect** bars under the **Connection** tab.

  It is in these two functions where the input to connect network to PolyScope X is applied.



7.3.1. Network

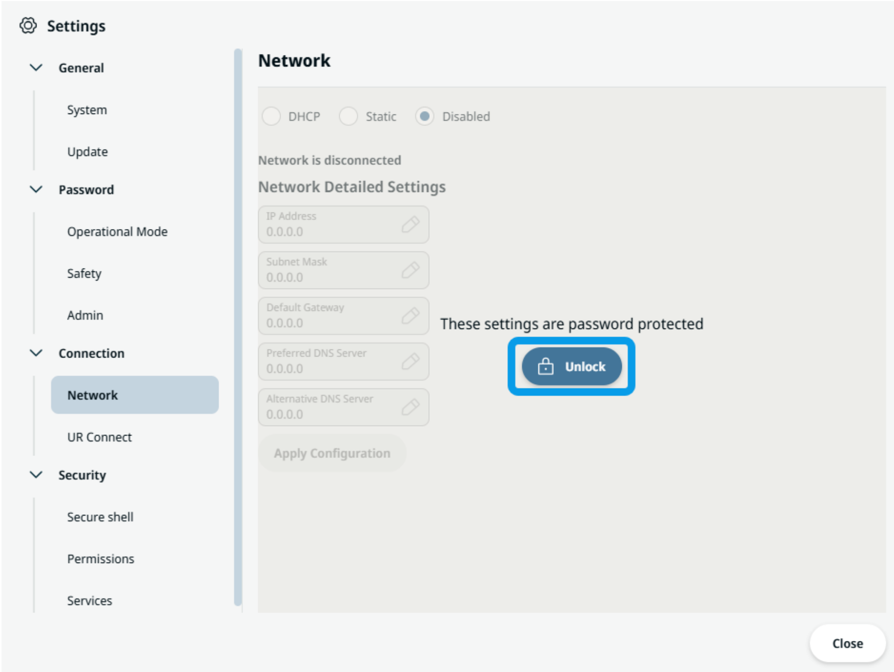
Description

The network settings are used to connect the PolyScope X to a secure network.

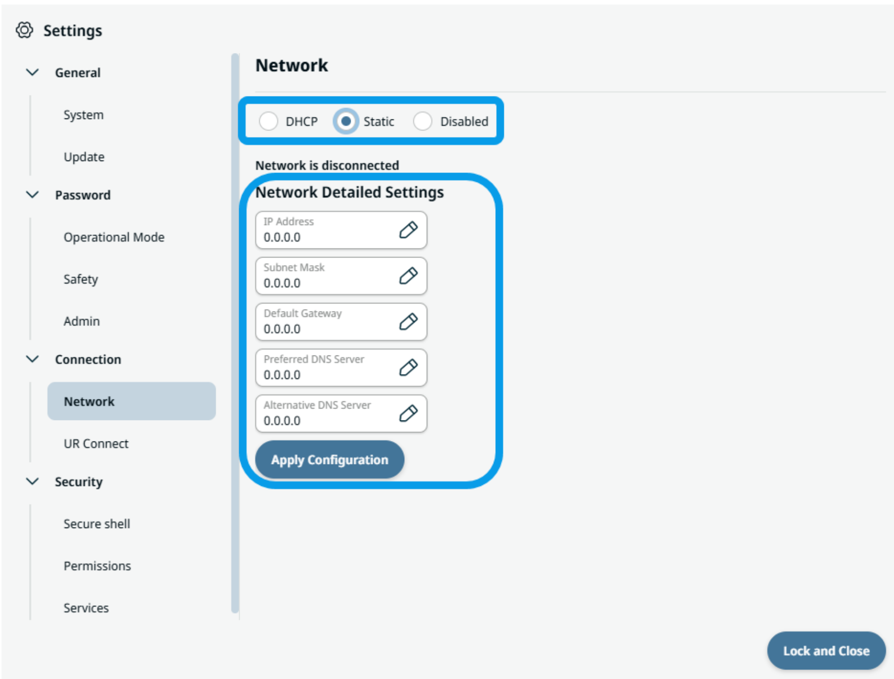
To connect via Network

In the **Network** screen:

1. Tap Unlock.



2. Type the admin password.
3. Choose your preferred Network choice and confirm. Your main screen shows you three Network choices.
4. Tap your preferred Network choice.
5. When **Static** is chosen, fill in the box of the five required fields of Network Detailed Settings.



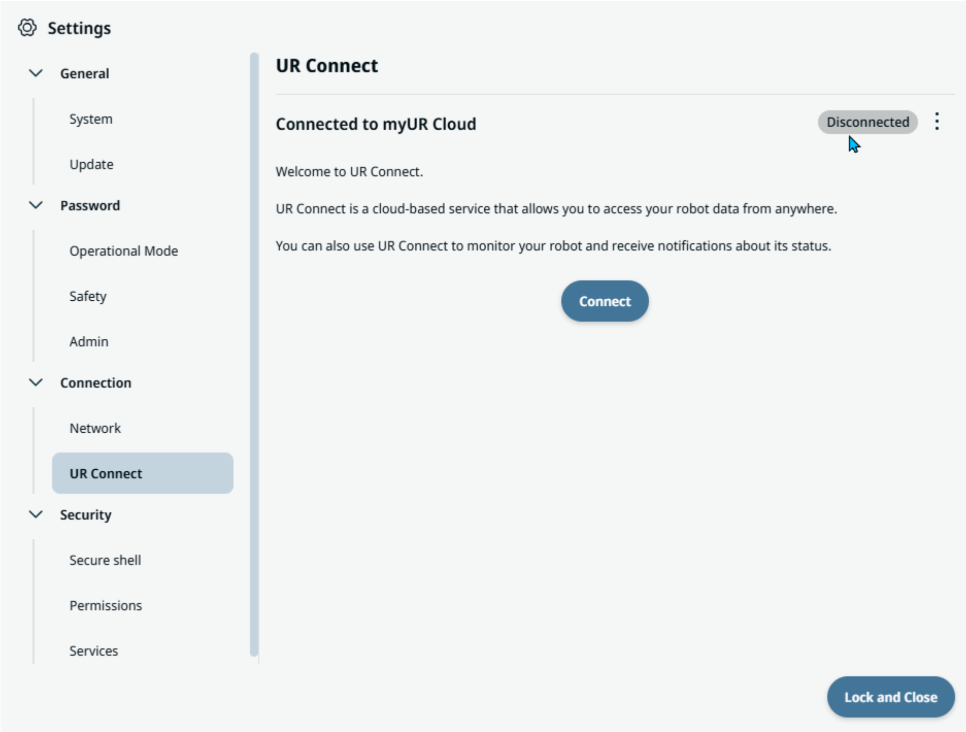
7. Tap Apply Configuration.

7.3.2. UR Connect

Description The UR Connect menu is used together with the UR Connectivity Kit.

To connect via UR Connect

In **UR Connect**:

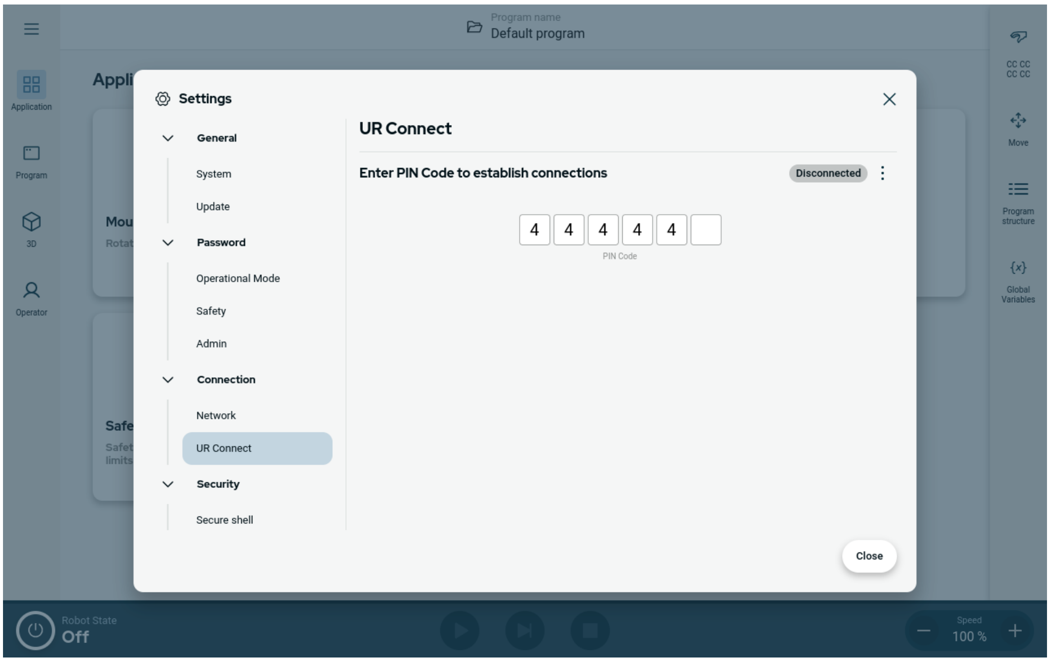


1. Tap Connect.
2. Type your six-digit PIN code.

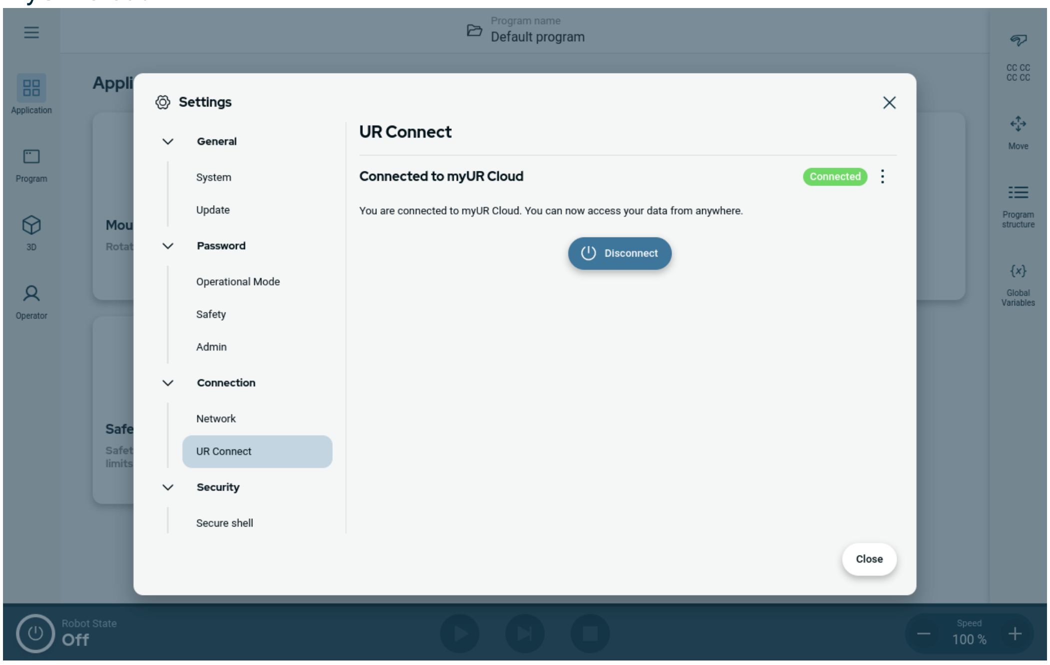
Connect PolyScope X to myUR Cloud

You have to connect your PolyScope X software to the myUR Cloud service. You need to find you PIN code in your myUR account.

1. Go to Settings.
2. Go to UR Connect.
3. Hit the "Connect" button on the main UR Connect page.
4. Add you pin code from myUR.

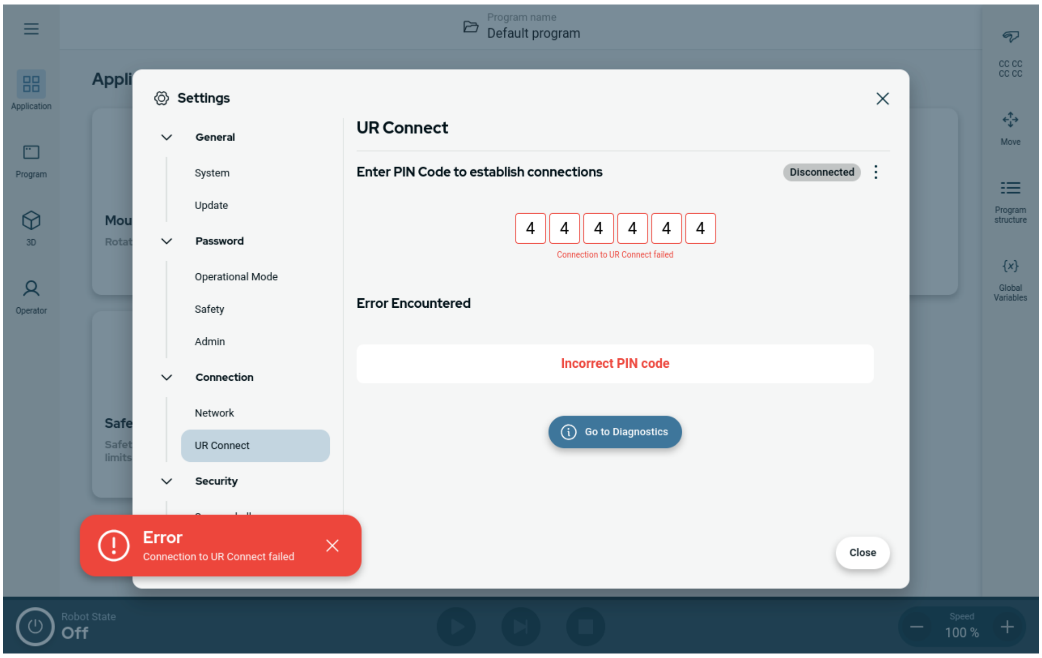


When you see the green icon in the right corner of the window, you are connected to the myUR Cloud.



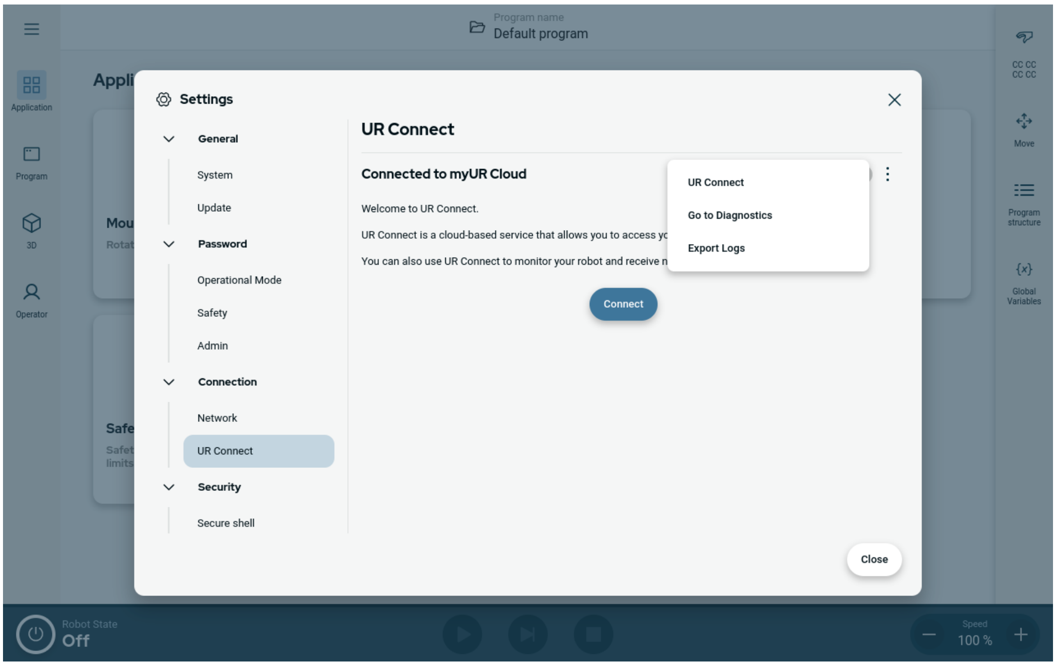
Unsuccessful connect

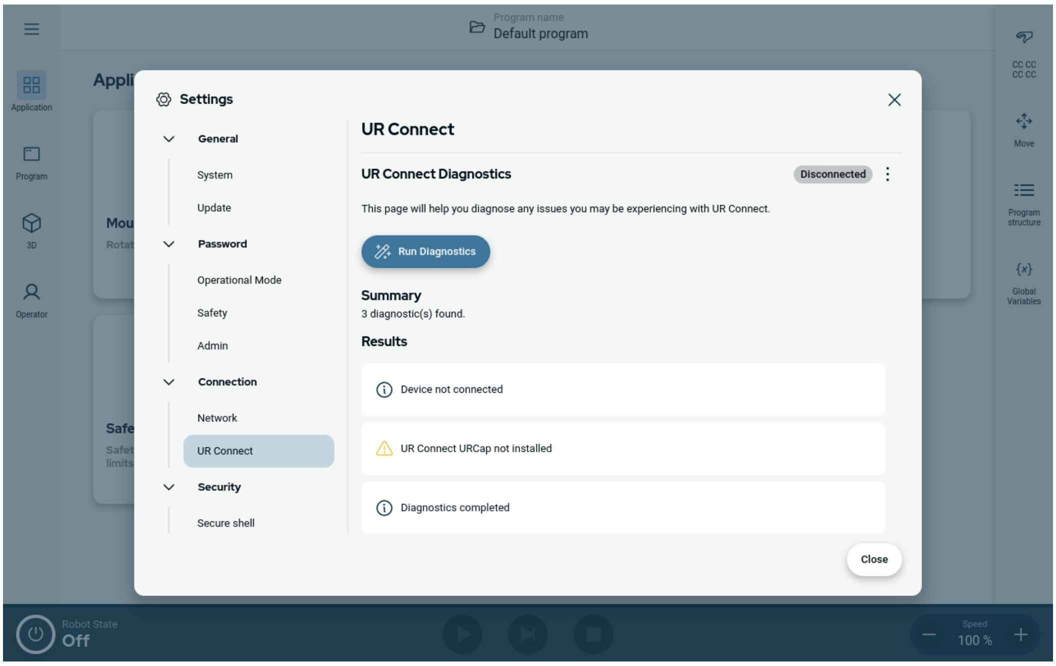
If you see the "Incorrect PIN code", please review your PIN code from myUR.



Diagnostics If you experience any unexpected when the UR Connect is active, you can go to the Diagnostics.

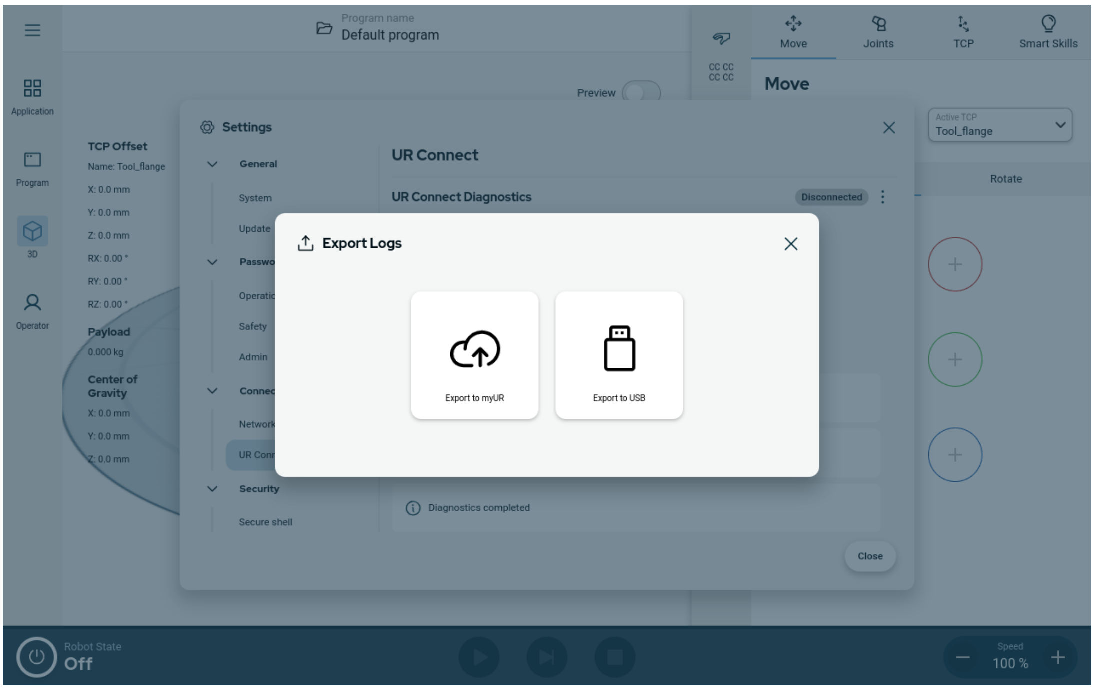
1. Go to Settings.
2. Go to UR Connect.
3. Hit the kebab menu in the top right corner.
4. Select the "Diagnostics".





Export logs It is possible to export the UR Connect logs from your PolyScope X software.

1. Go to Settings.
2. Go to UR Connect.
3. Hit the kebab menu in the top right corner.
4. Select the "Export Logs"
5. Select "Export to myUR" or "Export to USB".



7.4. Security

Description In the security settings, you can access the following:

- Secure Shell (SSH) Access
- Permissions
- Services

7.4.1. Secure Shell (SSH) Access

Description You can manage remote access to the robot using Secure shell (SSH). The Secure shell security settings screen allows administrators to enable or disable SSH access to the robot.

To enable/disable SSH

1. Access the Hamburger menu and select **Settings**.
2. Under Security, tap **Secure shell**.
3. Enter the password to **unlock** the settings.
4. Slide the **Enable SSH Access** to the on position.

To the far right of the Enable SSH Access toggle button, the screen shows the port used for SSH communication.

SSH Authentication

Authentication can occur with a password and/or with a pre-shared, authorized key. Security keys can be added by tapping the **Add Key** button and selecting a security key file. Available keys are listed together. Use the trash icon to remove a selected key from the list.

7.4.2. Permissions

Description

Access to the Networking, URCap Management and Updating PolyScope X screens is restricted by default, to prevent unauthorized changes to the system. You can change the permission settings to allow access to these screens. An Admin password is required to access Permissions.

To access Permissions

1. Access the Hamburger menu and select **Settings**.
2. Navigate to Security and tap **Permissions**.

Additional system permissions

You can also lock a few important screens/functionalities with the Admin password. On the Permissions screen in Security section in the Settings menu, it is possible to specify which additional screens are to be protected by the Admin password and which screens are available to all users. The following screens/functionalities can optionally be locked:

- Network settings
- Update settings
- URCaps section in the System Manager
- UR Connect

To enable/disable system permissions

1. Access Permission as previously described. The protected screens are listed under Permissions.
2. For the desired screen, slide the On/Off toggle switch to the On position to enable it.
3. To disable the desired screen, slide the On/Off toggle switch to the Off position.

The screen locks again once the toggle is in the Off position.

7.4.3. Services

Description

Services allow administrators to enable or disable remote access to the standard UR services running on the robot, such as Primary/Secondary Client interfaces, PROFINET, Ethernet/IP, ROS2, etc.

Use the Service screen to restrict remote access to the robot by only allowing external access to the services on the robot which the specific robot application is actually using. All services are disabled by default to provide maximum security. The communication ports for each service are to right of the On/Off toggle button in the list of services.

Enabling ROS2

When the ROS2 service is enabled on this screen, you can specify the ROS Domain ID (values 0-9). After changing the Domain ID, the system restarts to apply the change.

7.5. Unlock Safety Settings

Description

You have to unlock the safety settings to change them.

> **NOTICE**
> Safety Settings are password protected.

The default password for safety: **ursafe**.

Unlock safety settings

1. In the PolyScope X Main Navigation, tap the Application tab.
2. On the workcell screen tap the Safety icon.
3. Observe that the Robot Limits screen displays, but settings are inaccessible.
4. Enter the safety password and tap UNLOCK to make settings accessible.
5. Tap LOCK or navigate away from the Safety menu to lock all Safety item settings again.

8. Operational Mode

Description

You access and activate different modes using Teach Pendant or the Dashboard Server. If an external mode selector is integrated, it control the modes - not PolyScope or the Dashboard Server.

Automatic Mode When this mode is activated the robot can only execute a program of pre-defined tasks. You cannot modify or save programs and installations.

Manual Mode When this mode is activated you can program the robot. You can modify and save programs and installations. The speeds used in Manual Mode must be limited to prevent injury. When the robot is operating in Manual Mode, a person could be positioned within reach of the robot. The speed must be limited to the value that is appropriate for the application risk assessment.

> **WARNING**
> Injury can occur if the speed used, while the robot is operating in Manual Mode, is too high.

Recovery Mode This mode activates when a safety limit from the active limit set is violated, the robot arm performs a Stop Category 0. If an active safety limit, such as a joint position limit or a safety boundary, is violated already when the robot arm is powered on, it starts up in Recovery mode. This makes it possible to move the robot arm back within the safety limits. In Recovery mode, the movement of the robot arm is restricted by a fixed limit that you cannot customize.

High Speed Manual Mode When this mode is enabled, you can temporarily exceed the default speed limit of the tool and the elbow.
The robot performs a Safeguard Stop in Manual mode, if a Three-Position Enabling Device is configured, and either released (not pressed) or it is fully compressed.

Switching between Automatic mode to Manual mode requires the Three-Position Enabling Device to be fully released and pressed again to allow the robot to move. When using High Speed Manual Mode, use safety joint limits or safety planes to restrict the robot's moving space.

> **NOTICE**
> After five minutes of inactivity the speed limit resets to the default.

To enable High Speed Manual

1. Tap **Application** and select **Safety**.
2. Access the **Three Position** options.
3. On the page, slide the button **Allow manual high speed**.

Mode switching

Operational mode	Manual	Automatic
Move robot with +/- on Move Tab	x	
Freedrive	x	
Execute Programs	Reduced speed*	x
Edit & save program	x	

*If a Three-Position Enabling device is configured, the robot operates at Manual Reduced Speed unless High Speed Manual Mode is enabled.

WARNING

- Any suspended safeguards must be returned to full functionality before selecting Automatic Mode.
- Wherever possible, Manual Mode shall only be used with all persons located outside the safeguarded space.
- If an external mode selector is used, it must be placed outside the safeguarded space.
- No-one is to enter, or be within, the safeguarded space in Automatic Mode, unless safeguarding is used or the collaborative application is validated for power and force limiting (PFL).

Three-Position Enabling Device

When a Three-Position Enabling Device is used and the robot is in Manual Mode, movement requires pressing the Three-Position Enabling Device to the center-on position. The Three-Position Enabling Device has no effect in Automatic Mode.

NOTICE

- Some UR robot sizes might not be equipped with a Three-Position Enabling Device. If the risk assessment requires the enabling device, a 3PE Teach Pendant must be used.

A 3PE Teach Pendant (3PE TP) is recommended for programming. If another person can be within the safeguarded space when in Manual Mode, an additional device can be integrated and configured for the additional person's use.

Switching Modes

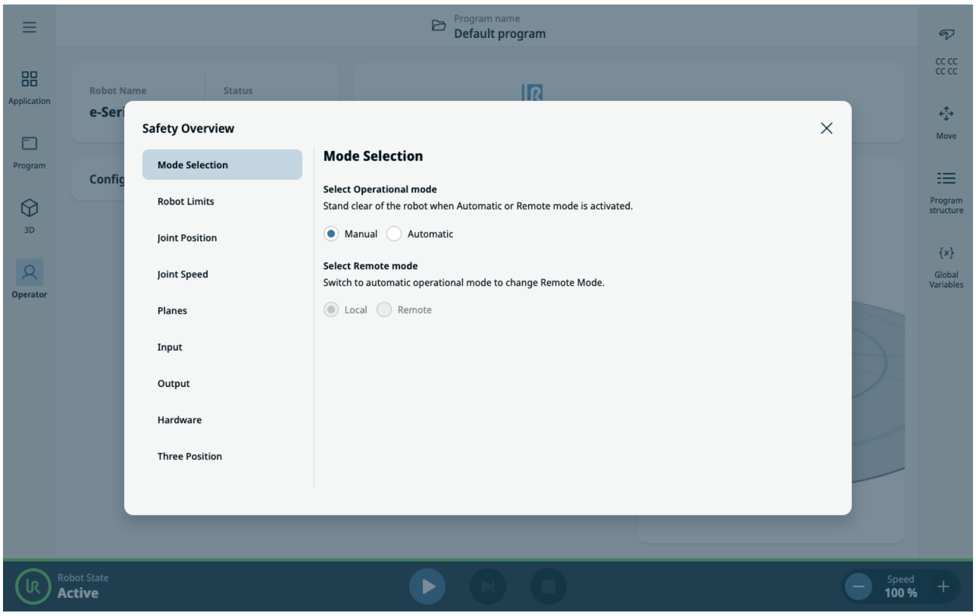
To switch between modes, in the Right Header, select the profile icon to display the Mode Selection.

- Automatic indicates the operational mode of the robot is set to Automatic.
- Manual indicates the operational mode of the robot is set to Manual.

PolyScope X is automatically in Manual Mode when the Safety I/O configuration with Three-Position Enabling Device is enabled.

Select Remote Mode

It is only possible to change the remote mode, when you have changed the operational mode to "Automatic".
If you change remote mode from "remote" to "local" the operational mode will go back to "manual".



8.1. Manual Operational Mode

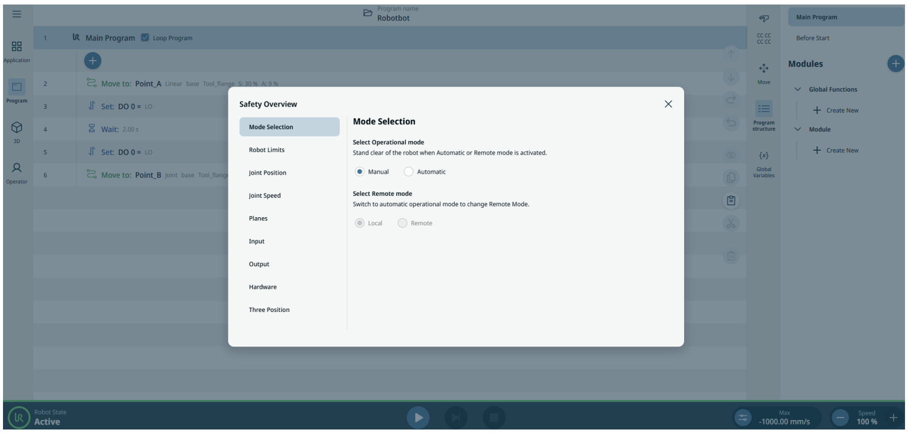
Description

In **Manual Mode**, you can create, modify, and save robot programs, applications, and installations. This has four properties:

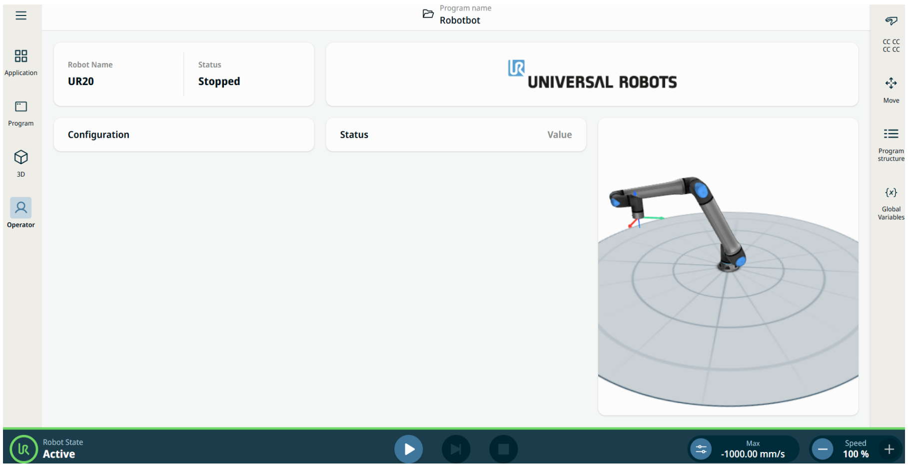
- A 3-position enabling (3PE) device must be held in its center position.
- Modify and save programs.
- Reduced speed control to 250 mm/s.
- Manual high speed is enabled by adjusting the **Manual High Speed** slider in the footer. A confirming action is required before higher speeds can be achieved. If the enabling device has been released for five minutes, manual high speed must be reenabled before higher speeds than 250 mm/s can be obtained.

To use Manual mode

1. Follow the steps of “To access Mode Selection” in Operational Mode. See Operational Mode.



Manual mode is the default setting. The main screen contains all the functionalities of the PolyScope X interface.



8.2. Automatic Operational Mode

Description

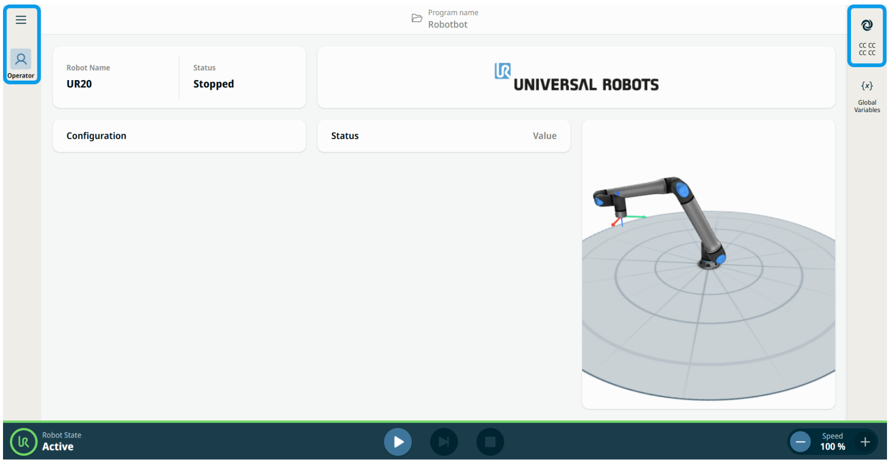
In **Automatic Mode**, the robot can only execute a program of predefined tasks. You cannot modify or save programs and installations. This has two properties:

- Automatic program execution is allowed
- Motions at full speed (complying with safety limits)

To use Automatic mode

1. Follow the steps of “Switching Modes” in Operational Mode. See Operational Mode.
2. Tick the **Automatic** button in the Select Operational mode, enter the operational mode password to unlock, and Confirm.

   While in automatic mode, the PolyScope X interface is in the operator screen and the tabs in main navigation and sidebar are reduced. Additionally, the automatic mode icon  is displayed above the Safety checksum .

3. You can tap the **Play button** in the footer or the robot control bar.

9. Remote Control

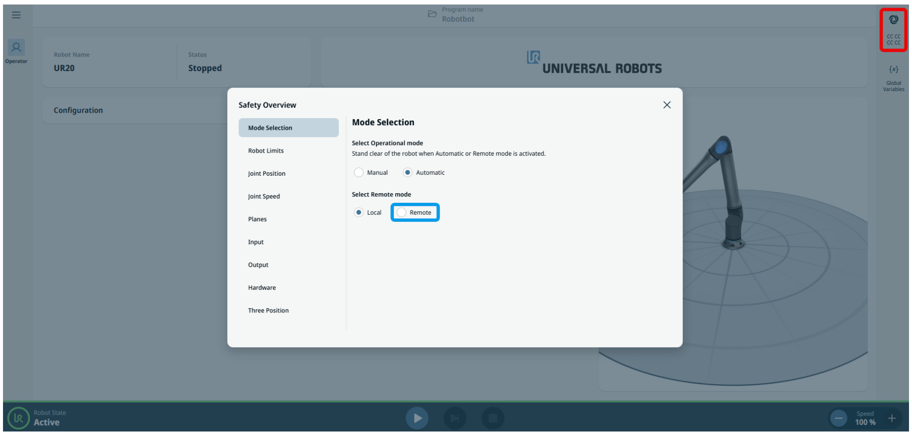
Description

Remote Control allows you to control the robots via external sources, such as controller sockets, IOs, and the dashboard server. The other control is via the teach pendant (TP), called local control.
Remote control can be used to send simple commands to PolyScope X such as starting or loading programs, as well as sending UR Script commands directly to the controller.

> **CAUTION**
> To ensure safe usage, the robot can either be in Remote Control mode or Local Control mode.
> Local Control mode will ensure that commands sent to the controller from an external source will be rejected while the robot is controlled in person.

To enable Remote Control

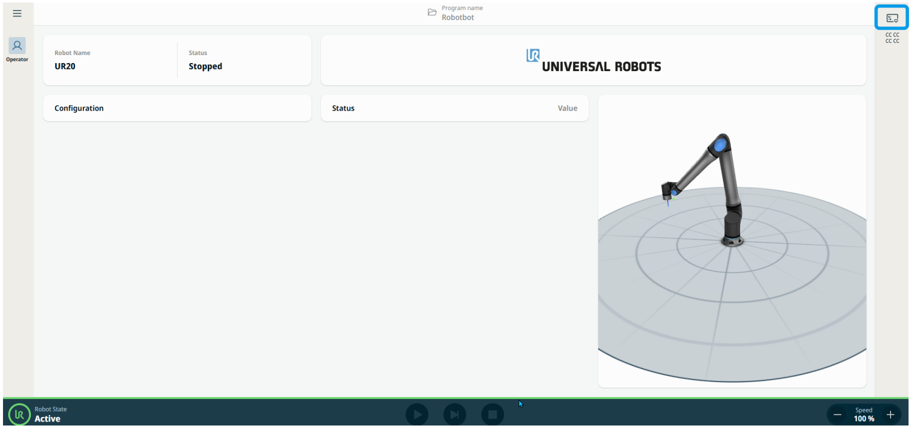
1. Follow the steps to switching into Automatic operational mode. See Automatic Operational Mode.
2. Tap the Safety Overview/Checksum icon. The Safety Overview screen appears.



3. In the **Select Remote mode**, tick the Remote button.

> **NOTICE**
> Remote Mode is only active when the application is in Automatic mode.

4. The **Remote Mode icon** is displayed above the Safety checksum to clearly indicate the system is under remote supervision. All editing and control actions are disabled, and only the Operator Screen remains accessible in view-only mode.



> **NOTICE**
> - Although Remote Control limits your actions in PolyScope, you can still monitor robot state.
> - When a robot system is powered off in Remote Control, it starts up in Remote Control.

Local Control does not allow	• Power on and brake release sent to the robot over network • Receiving and executing robot programs and installation sent to the robot over network • Autostart of programs at boot, controlled from digital inputs • Auto brake release at boot, controlled from digital inputs • Start of programs, controlled from digital inputs
Remote Control does not allow	• Moving the robot from Move Tab • Starting from Teach Pendant • Load programs and installations from the Teach Pendant • Freedrive

10. Robot-API

Description The Robot-API provides a RESTful interface for remotely operating the robot. It follows a predictable, resource-oriented URL structure, accepts form-encoded request bodies, returns JSON-formatted responses, and adheres to standard HTTP response codes for clear and consistent communication.

Network To interact with the Robot-API, the robot must be connected to a network. The API documentation is always accessible directly from the robot itself and includes comprehensive guidance on how to utilize the available endpoints.
For more details on network setup, please see Network in the PolyScope X Software Handbook.

API

Environment	API Base URL	Documentation URL
Robot	http://[ip]/universal-robots/robot-api	http://[ip]/universal-robots/robot-api/docs
UR Simulator	http://localhost/universal-robots/robot-api	http://localhost/universal-robots/robot-api/docs

11. System Manager

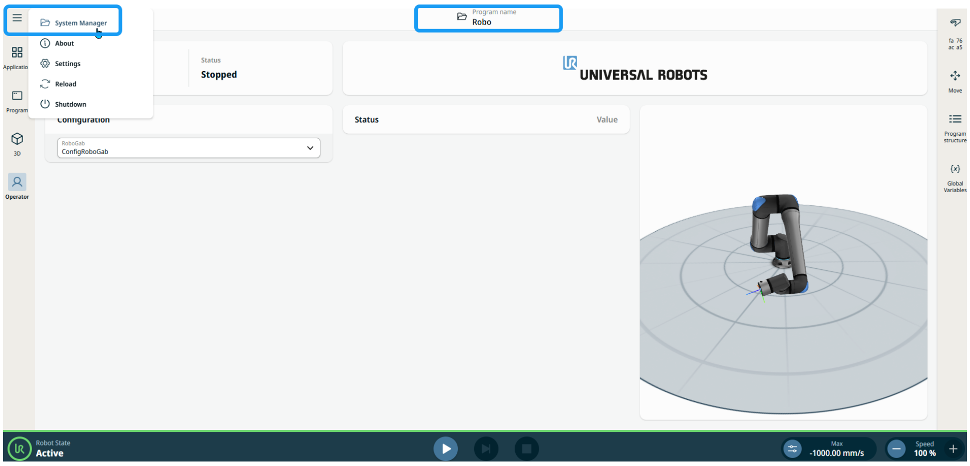
Description

The **System Manager** enables you to do the following functions:

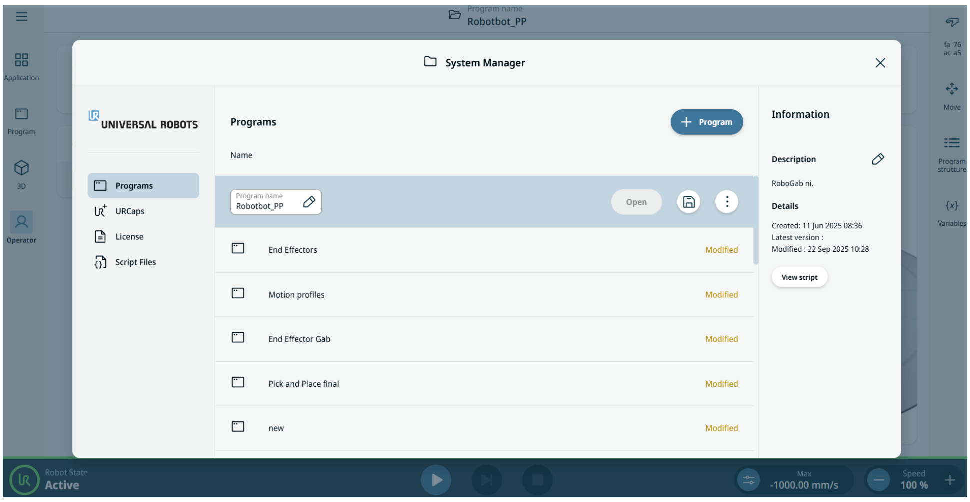
- See all created programs and installed URCaps
- Create a new program
- Import programs
- View script of selected program
- Write a program description

To access System Manager

There are two ways to access the **System Manager**:
Header: Tap the header, which contains solely the **Program name** menu.
Hamburger Icon: In the main navigation, tap the hamburger icon and then System Manager.



The **System Manager** screen appears, which contains the following components located in three columns.



On the left column are found the following sections:

- Programs
- URCaps
- License
- Script Files

On the center column are found the **Program Manager**, which lists the created program, and the **+ Program** button/icon, which enables the creation of new program and importing a program.
On the right column are found the editable **Description** field and **View script** button.

11.1. Robot Registration and License File

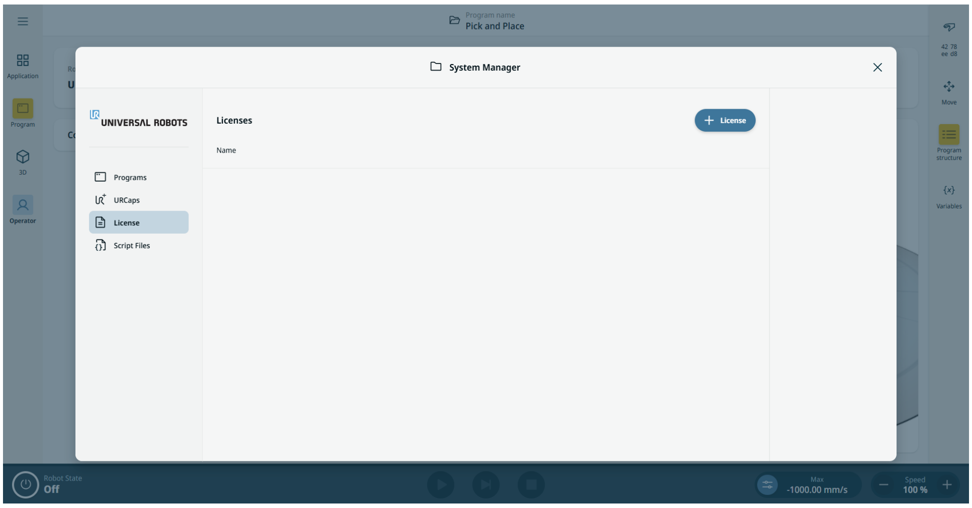
Description

It is necessary to register the robot and download and install the License File, because the license file will include all available software licenses.

Activate the Software Licenses

> **NOTICE**
>
> If you have more than one active license, all licenses will be included in the downloaded license file.

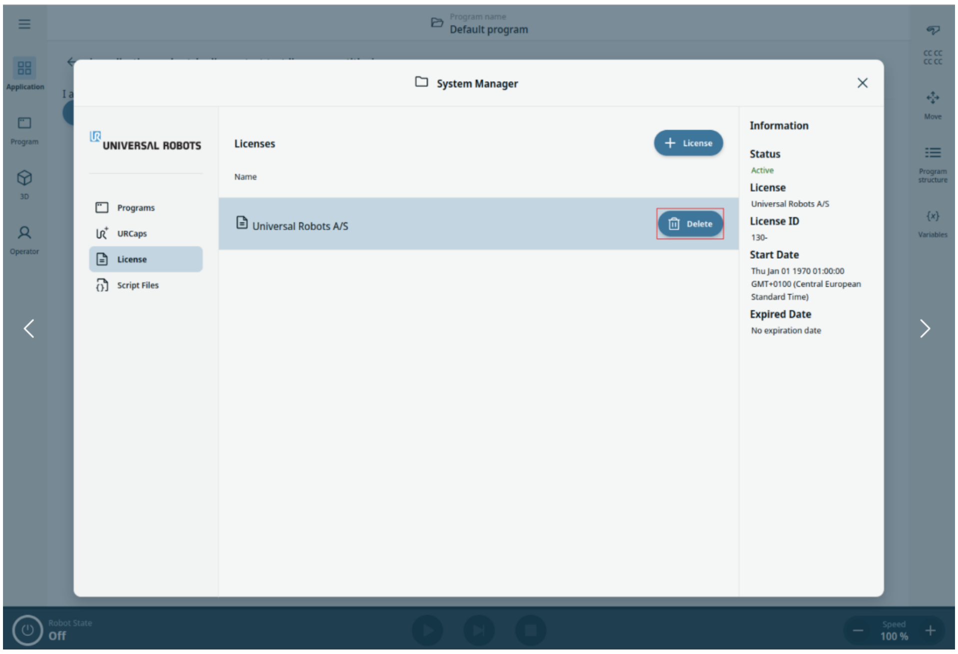
1. Download the license file to your PC.
2. Copy the license file to the USB and connect it to the Teach Pendant.
3. On the Main navigation, tap the **Hamburger menu** and select **System Manager** to display content.
4. In the list, select the **License** and tap **+License**.
5. Select the license file saved and tap **Open** to confirm robot registration.
6. On the bottom left, tap **Exit**.



Deactivate Software Licenses

- A new license file is required if the robot changes owners. In this case, the license file must be deactivated.
- If you purchase a new software license for your robot, it is necessary to deactivate and reactivate the license file in order to add the new software license.

1. In the Main navigation, tap the **Hamburger Menu** and select **System Manager**.
2. In the list, select **Licenses**.
3. Select the license to deactivate/remove and tap **Delete**.



11.2. URCaps

Description

You can manage your existing **URCaps** or install a new one in your robot.

To manage URCaps

1. In the Header, press the Hamburger menu and select **Settings**.
2. Under System Manager, select **URCaps**.
3. Tap the **+** button, select the **.urcap** file and press **Open**.
4. If you wish to proceed with the installation of that URCap, press **Restart**. After that step, the URCaps is installed and ready to be used.
5. To remove an installed URCaps, select it from Active URCaps, press the **-** button and press **Restart** so changes can take effect.

Active URCaps

Details about the new URCap appear in the **Active URCaps** field.
A status icon indicates the state of the URCap, as listed below:

- URCap ok: The URCap is installed and running normally.
- URCap fault: The URCap is installed but unable to start. Contact the URCap developer.
- URCap restart needed: The URCap has just been installed and a restart is required.

Example

Error messages and information about the URCap appear in the **URCaps Information** field. Different error messages appear depending on the type of error/s detected.

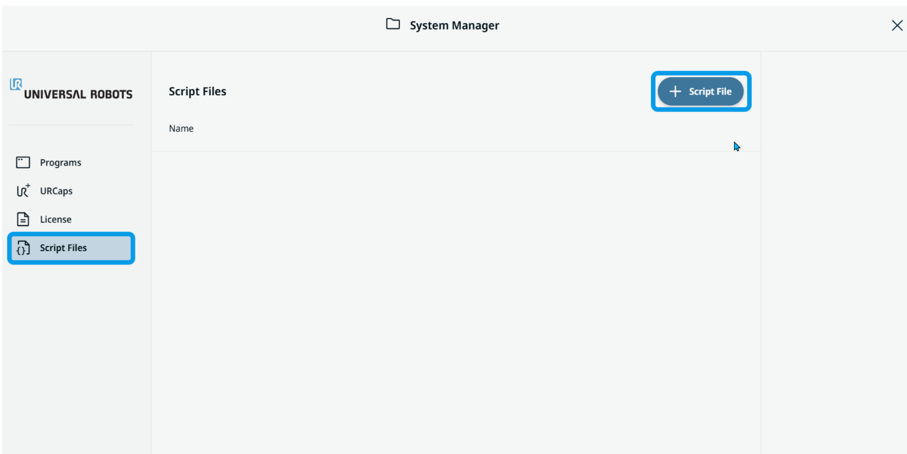
11.3. Script Files

Description

Script files are used in the program through the Script Node, imported from a disk into PolyScope X. The script files are managed via the System Manager

To access Script Files

1. Navigate to the **System Manager** by tapping either the System Manager or header.
2. Tap **Script Files** in the left column, and add your script file **+ Script File**. The file on disk should have the **.script** extension.

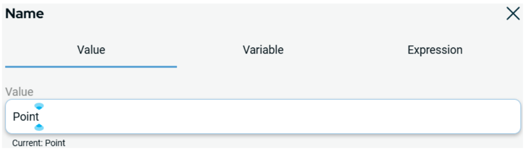


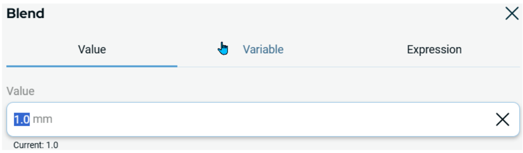
Script files are kept according to their file name. So reimporting the script file again from disk will overwrite the old one. Once imported, the script file can be deleted or viewed.

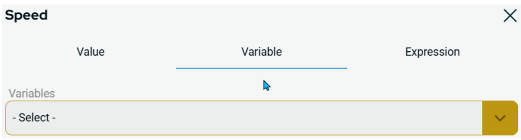
12. Value-Variable-Expression Tabs

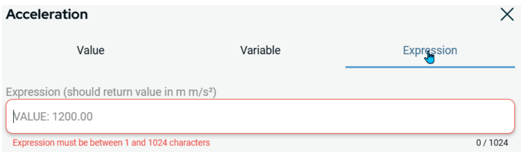
Description

The **Value**, **Variable**, **Expression** tabs appear as pop-up dialog in opening commands. They are common components used in almost all commands.









Value tab description

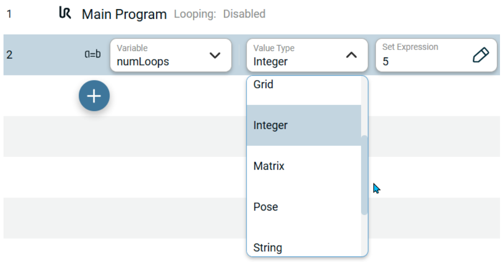
The **Value** tab is used when you know the exact fixed value of a program. It restricts you to values appropriate for the given variable.
The **Value Type** drop-down tab in the **Assignment** command contains the type of variable you are creating and has one of the following values:

Type	Allowed Values/Examples	Examples
Array	A list of values	
Boolean	True/False	True
Float	A decimal point value	1.23
Grid	A grid defined in the application	
Integer	A whole number	5
Matrix		
Pose	A pose for the robot to move	p[0,0,0,0,0,0,]
String	Text	“Hello World”
Waypoint	A waypoint is defined in the program	

Example use of Value tab

Create a restrictive value appropriate for a given variable, which runs five times.

1. In the left toolbar, tap the **Program** icon.
2. Tap the **add** icon.
3. Tap the **Assignment** node command. See Assignment.
4. On the **Variable** field, tap **Rename**.
5. Rename to **numLoops**.
6. Tap **Confirm**.
7. Set **Value Type** to **Integer**.
8. Tap **Set Expression** and enter **5** in the **Value** tab.
9. Tap **Confirm**.



Note: As your variable is integer, the value only allows you to enter whole numbers.

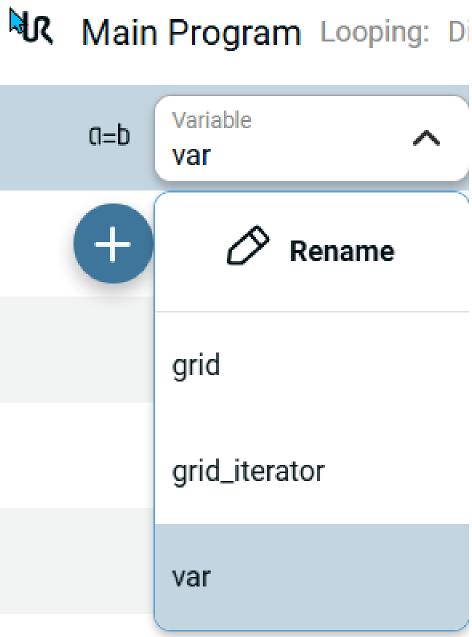
Variable tab description

The **Variable** tab allows you to pick from a list of variables defined in the program or application. Only variables of the appropriate type is shown. When you set the value of a digital output, only Boolean variable type is shown. When you set the value of an analog output, only Integer or Float variable type is shown.

Example use of Variable tab

The **Variable** drop-down field contains the name of the variable. By default, the variable is **var**.

1. Got to steps 1-3 in the use of Value tab.



   The tab contains the list of variables already in the program, such as from another Assignment node, Joint Move or Linear Move nodes, and other application nodes, grid iterators.

2. Tap **Rename** to change the name of the created variable.
3. To iterate and add options, repeat the process.

12.1. Expression Editor

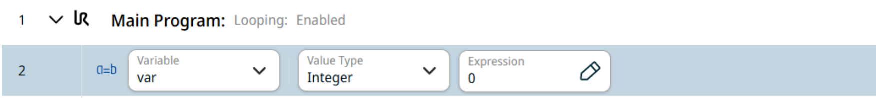
Description

The **Expression** tab enables you to insert an expression, which should be a valid URScript and evaluates to the correct type. An expression in an **If** node evaluates to a Boolean. Whereas a String type requires enclosing quotes to be used. Furthermore, the **Expression** tab allows you to easily insert a variable into the expression.

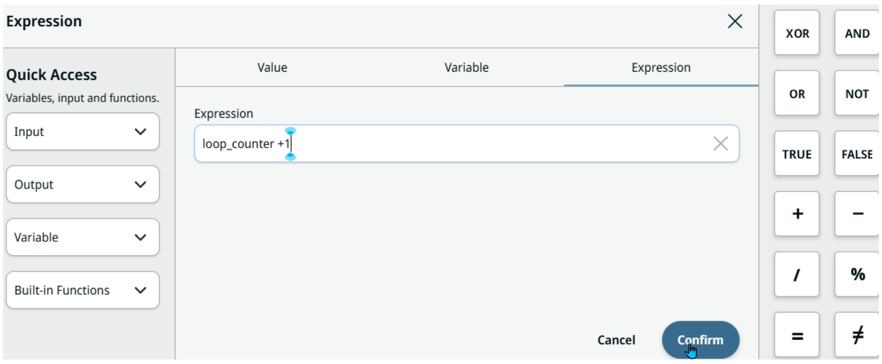
Example use of Expression tab

Create a program loop counter.

1. In the main navigation, tap the **Program** icon.
2. Tap the **add** icon.
3. Tap the **Assignment** node command. See Assignment.
4. Select **Integer** in the **Value Type** field.
5. Tap the **Expression** field.

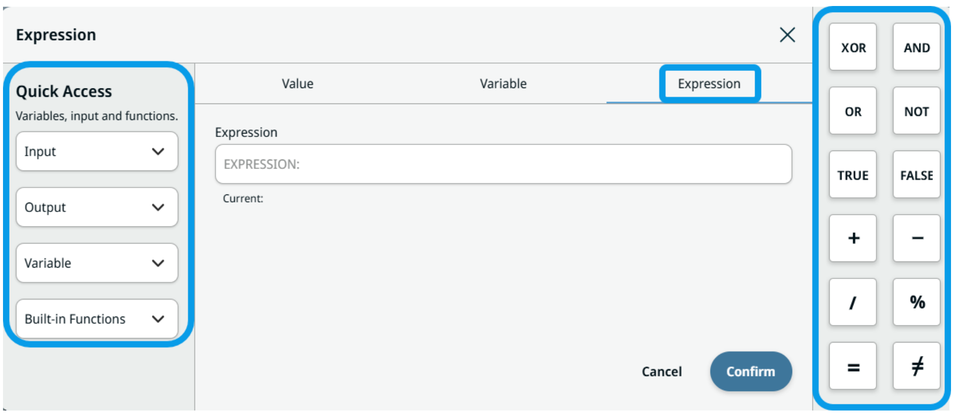


6. Tap the pop-up **Expression** tab, and the **Expression Editor** appears. Here you can set the expression to **loop_counter +1** and **Confirm**.



Expression Editor

In the Expression Editor, you can find shortcuts for inserting functions, IOs, and logic operators into expressions.
On the **left column**, you see the Quick Access section that allows you to easily insert an Input, Output, Variable, or Built-in Function at the current cursor position.
On the **right column** are found buttons to insert common logic operators and values, such as AND, OR, NOT, TRUE, FALSE. This functionality has been added to every tabbed into dialog, as well as to few node fields that previously only accepted a string, such as the **If** node.



13. OptiMove

Description

The **OptiMove** motion value is an advanced feature of PolyScope X **Joint and Linear Move** commands. See 16.3.15 Joint Move on page 156 and 16.3.16 Linear Move on page 163.

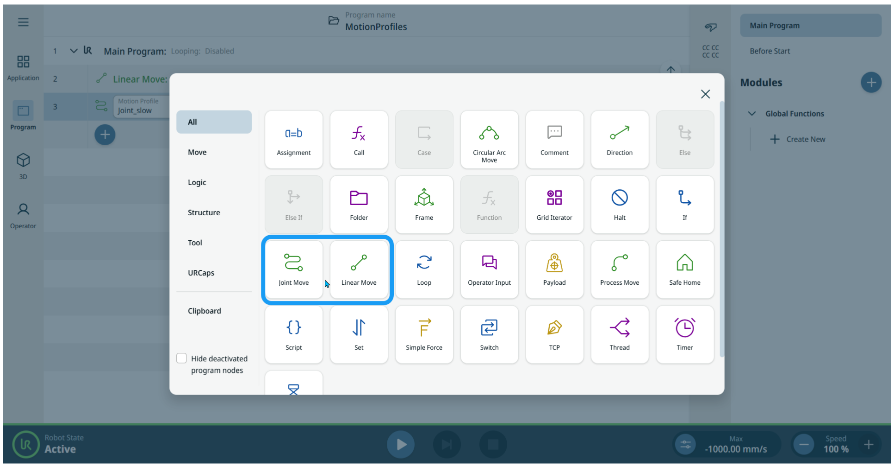
OptiMove allows you to have the following benefits:

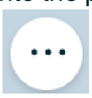
- More dynamic robot acceleration and deceleration
- Smoother moves with less jerk
- More fluent acceleration and deceleration
- Higher travel speeds
- Reduced vibrations and loads on the robot and mounted equipment

The combined effect in **OptiMove** generally results in improved cycle time with more fluent motion and reduced strain on the robot.

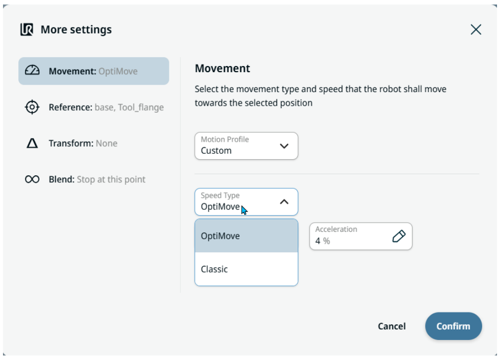
To access OptiMove

1. Go to the main program screen. See Command Nodes.
2. Tap the **Joint Move** or **Linear Move** icon on the Commands toolbox.



A node is inserted into the program tree. The node has two editable fields and tabs, and an ellipsis icon .

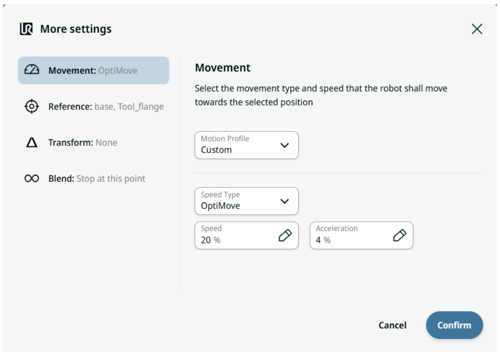
3. Tap the ellipsis icon and the **More settings** screen appears.



4. Tap **Movement** on the left side of the screen.

   On the right side, two tabs and fields appear:

   - Motion Profile
   - Speed Type
   - Speed tab
   - Acceleration tab



5. In the **Motion Value** field, **OptiMove** is the default option. Choose **Classic** movement if you want to input speed and acceleration in degrees per second or mm per second.
6. Tap **OptiMove**.

   OptiMove settings are specified in percent to simplify the usage and setup.
7. In the **Speed** tab, enter the percentage of your preferred speed and tap **Confirm**.

   Speed tab sets the target travel speed as a percentage of the robot's maximum speed capability.
8. In the **Acceleration** tab, enter the percentage of your preferred acceleration and tap **Confirm**.

   Acceleration tab sets the target torque limits during acceleration and deceleration as a percentage of the robot's maximum power.

14. Smart Skills

Description

Smart skills are quick-access robot actions that you can start and stop to achieve a smoother and faster programming experience. Through these actions, you can manipulate the robot, any external systems, or end effectors.
These are the Smart Skills to choose:

Activated

- 14.1 Align to Plane on page 105
- 14.2 Align Z to Nearest Axis on page 105
- 14.3 Center on page 106
- 14.4 Freedrive on page 106
- 14.6 Move Into Contact on page 107
- 14.7 Retract on page 107
- 14.5 Home on page 107

Deactivated

- Put into Box
- Custom

To access Smart Skills

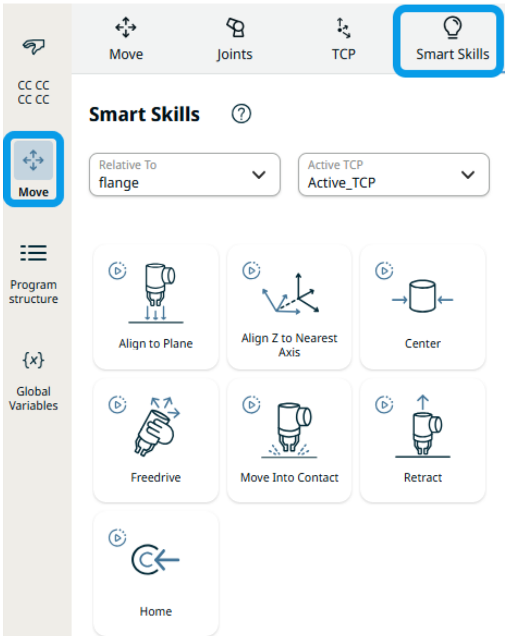
There are two ways to access Smart Skills:

- via **Move** section in sidebar
- via **Application** tab in the main navigation

Access via Move section

Via Move section is where you press and hold the button on the teach pendant while the smart skill is executing. The corresponding robot movement is seen in the main screen.

1. Tap the Move icon in the sidebar.
2. Choose the Smart Skills icon at the top right.



3. In the **Relative To** field, you can choose from the four features or frames or default coordinate systems:
   - **World**. This frame is the same as the base when the robot is mounted in a fixed location. The world frame is always fixed even when the robot is mounted in a moveable location, and you can update the base frame as the robot moves.
   - **Base**. The robot's base element, referring to the frame of the base, which is located in the center of base bottom. If your robot is mounted in a fixed location, this frame will never need to change, and other fixed locations can be defined relative to it.
   - **Flange**. It's placed on the robot's tool flange and can only be moved by moving the robot.
   - **TCP**. The position of the active TCP and updates as the robot moves when jogged or while a program is running.

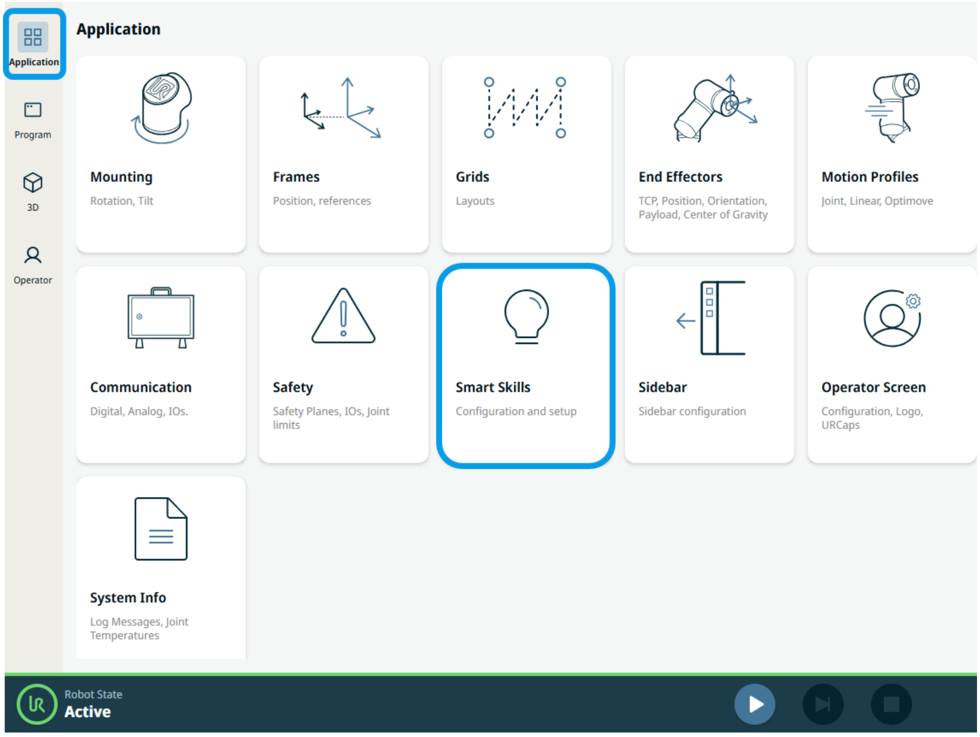
     See Frames to know more when to use the pertinent default coordinate system.
4. In the **Robot** field, you choose either **Active_TCP** or **Tool_flange**. If Active_TCP is selected, the name of the current active TCP is displayed.

Access via Application tab

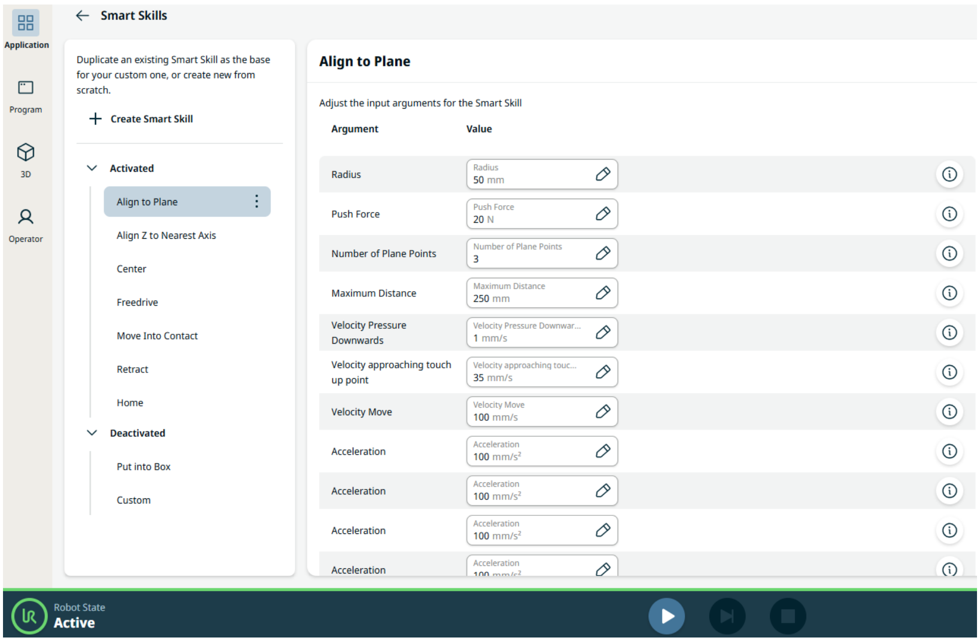
Via Application tab is where you can configure and set up the input arguments.

1. Tap the Application tab in the main navigation. See Smart Skills Application.

   Choose the Smart Skills application menu.



2. You are directed to the Smart Skills application screen.



14.1. Align to Plane

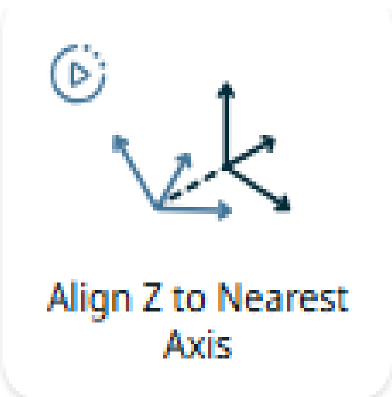
Description

Align to Plane assists you in orienting the TCP to a surface. While executing this smart skill, the robot will touch up 3 (default value) points on the surface from which it computes the plane's orientation. The contact points are laid out in a circular pattern which has a default diameter of 2.5 mm. This smart skill is especially useful for creating new frames in the Frames application.

14.2. Align Z to Nearest Axis

Description

Align Z to Nearest Axis finds the nearest axis of a frame and align the TCP z-axis to that. This functionality is used to align the robot's tool to already existing frames. You can change which frame the robot aligns to by changing the frame in the **Relative To** field.



Example

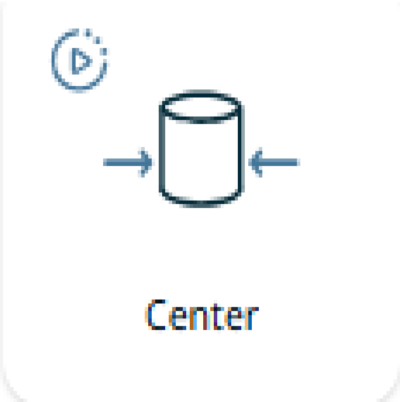
If you have a gripper and want to pick up an item, use this smart skill in order to have the robot actually move around in a predefined pattern in order to figure out where is the item positioned and what is the center of that. In this way, you can then center the robot.

14.3. Center

Description

Center is a dual functionality smart skill used to:

- Center the tool to a part in a fixture
- Center an object to a whole, given that the robot is already holding the object in its tool



14.4. Freedrive

Description

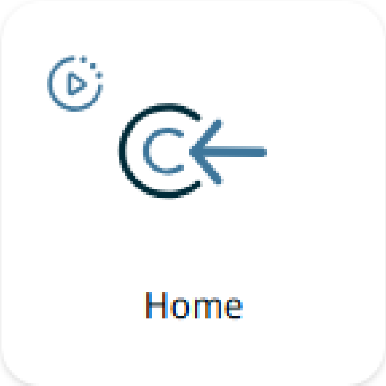
Freedrive smart skill enables you to set the robot program in a freedrive position.

Freedrive allows the robot arm to be manually pulled into desired positions and/or poses.

14.5. Home

Description

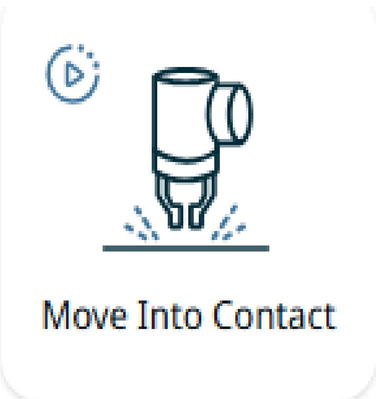
The **Home** Smart Skill moves the robot to a predefined home position. Through this, you can define a home position to easily go to a predefined posture from the Smart Skills tab. The upright zero position will be the default but can be changed in the Smart Skills Application tab, and it is available as a waypoint.



14.6. Move Into Contact

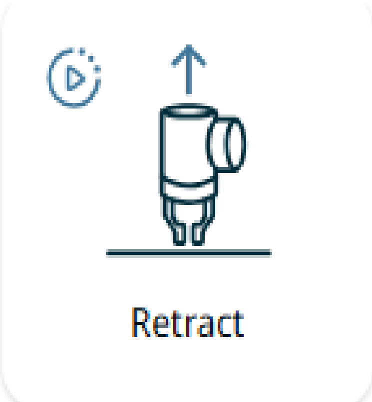
Description

The **Move Into Contact** smart skill enables you to move the robot in the TCP z-direction until it reaches a contact point. After reaching this contact point, the robot will retract a given distance. By default, this is set to 1 mm retraction distance.



14.7. Retract

Description	**Retract** smart skill moves the robot in the negative TCP z-direction.



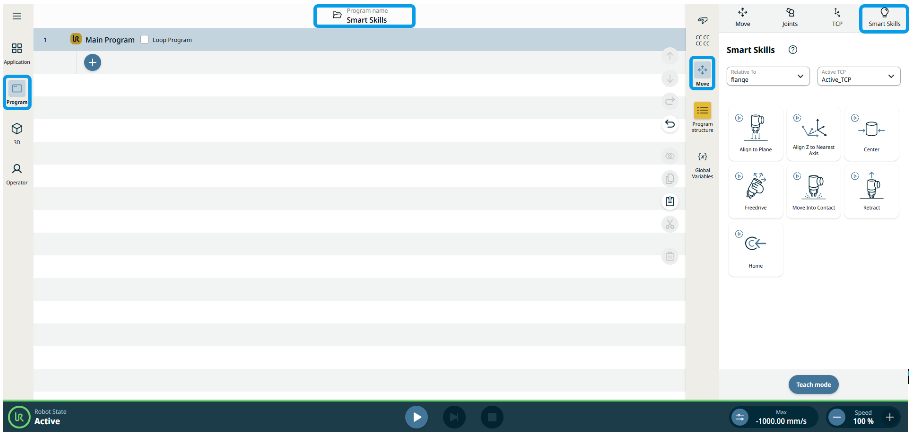
15. Teach Mode

Description

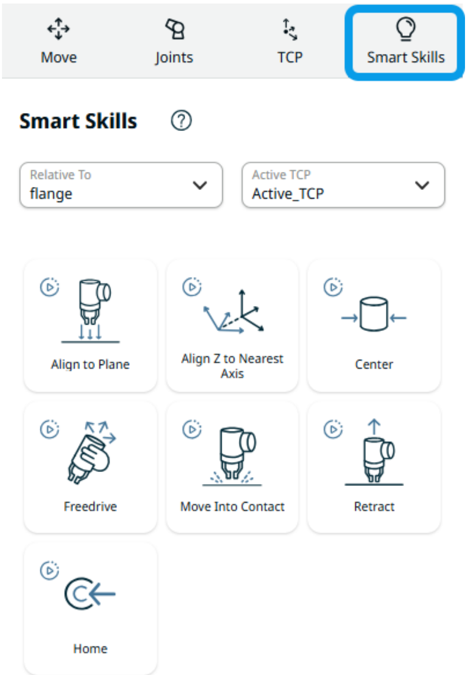
Teach Mode is a new way to program the robot by demonstration through teaching the robot what to do with a combination of freedrive and Smart Skills. Every action that is done in Teach Mode will be recorded and converted to program nodes.
To enable Teach Mode, navigate to the Smart Skills pane located in the Move drawer. Apply a Smart Skill or press the physical freedrive button to generate a program node in the tree. Additionally, you can undo the Smart Skill to restore the robot's previous state, for example, by returning it to its position before the action was performed. Read related topics Program and Smart Skills.

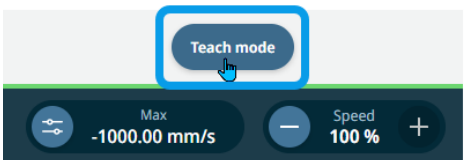
To access Teach Mode

1. Tap **Program** in the main navigation.
2. In the header, tap **Program Name** to use an existing program or create a new one where you want to apply Smart Skills program or freedrive.
3. Tap **Move** in the sidebar to expand the multitask screen, and you can see the **Smart Skills** pane in the top right part.

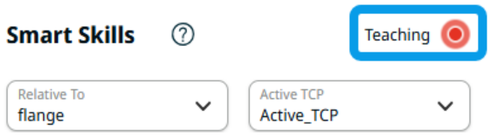


4. Tap Smart Skills , and you can see these components making up the Smart Skills screen:
    - Relative to field
    - Active TCP field
    - Align to Plane
    - Align Z to Nearest Axis
    - Center
    - Freedrive
    - Move Into Contact
    - Retract
    - Home
    - Teach Mode button



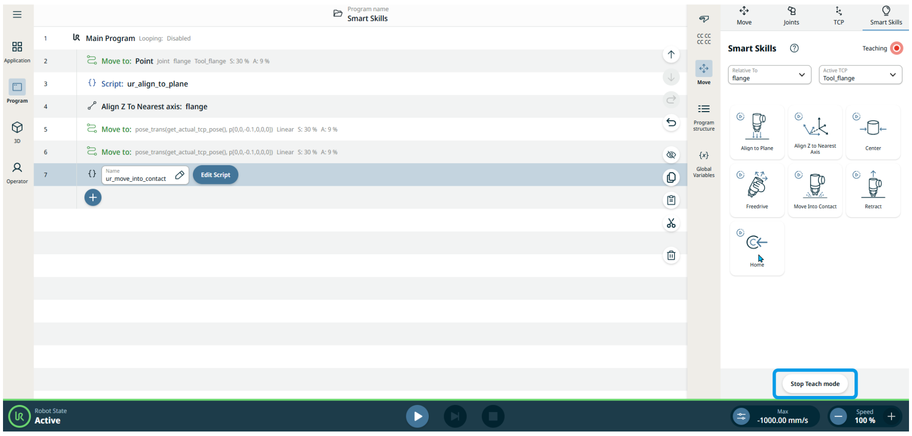


5. Tap the **Teach Mode** button, and you see a blinking red point indicating **Teaching**.



6. Tap a smart skill to start teaching, and tap it a second time to stop teaching. A configurable node is inserted into the program.
7. Continue with other Smart Skills.
8. Tap the **Stop Teach Mode** button when you are done.

   This is how the program tree looks like when, for example, you use three smart skills.



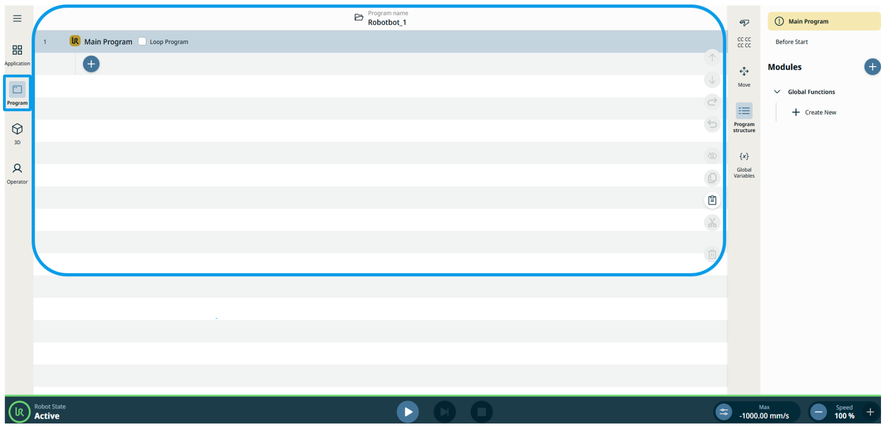
16. Program

Description

The **Program** menu enables you to create and modify programs for the robot.

To access Main Program screen

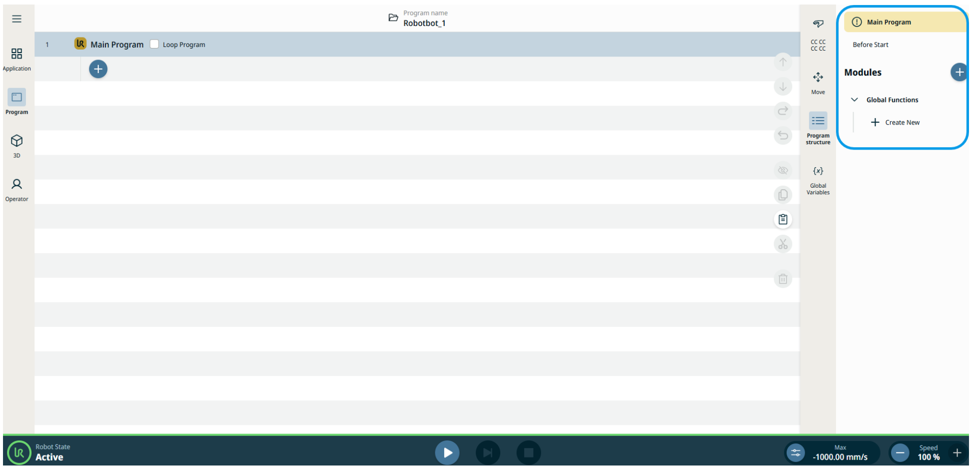
1. Tap the Program icon  on the main navigation.
2. You can see the **Main Program** setting in the main screen.



3. Tap the add icon  . The command and clipboard toolboxes appear. The command toolbox is the default setting.
4. In the commands toolbox, tap the command node you want to use.
5. Tap the clipboard toolbox to view your history of program creation.

To access Main Program control tabs

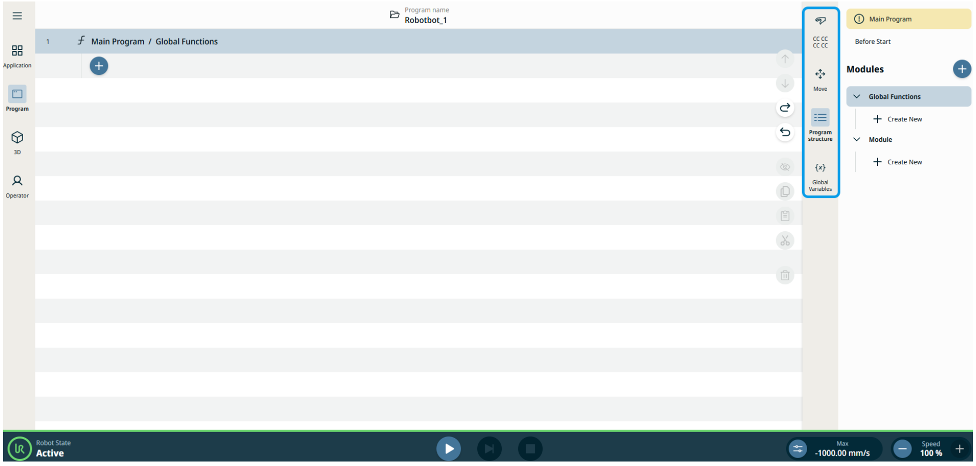
You can see the control tabs of the Main Program at the right side of the screen.



1. Tap the two tabs above **Modules**, and you can see the corresponding responses in the Main Program setting.
2. Tap the Modules **add icon** to create a module.
3. Under Global Functions, tap **+ create new** to add a new function.

To access Sidebar

The Sidebar contains **Safety Checksum**, **Move**, **Program structure**, and **Global variables**.



1. Tap  to access **Safety Checksum**.
2. Tap **Move** to program robot movement, joints, and rotation.
3. Tap **Program structure** to show or hide the control tab of the Main Program setting.
4. Tap **Global Variables** to view variables of global nature.

16.1. Create Program

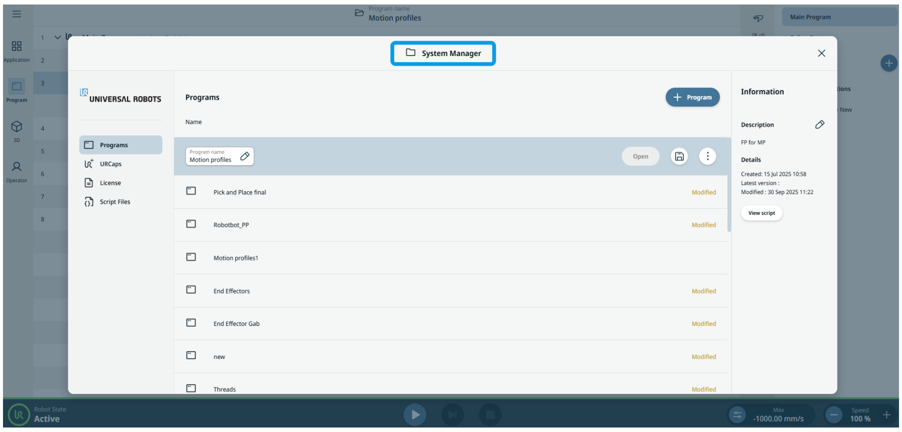
Description

The **Create** program command enables you to create and add programs for the robot.

To open Create program menu

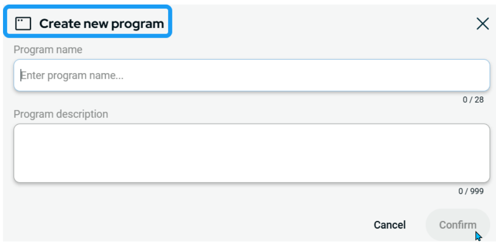
1. Tap the **Operator** icon  on the left toolbar.
2. Tap the **Program name** tab found topmost in the main screen.

   The **System Manager** screen appears.

3. Tap the add program icon. 

4. Tap **Create New Program**.

   A new box appears with the heading **Create new program**.
5. Enter your program name on the first box, **Program name**.
6. Enter the program description on the second box, **Program description**.
7. Tap **Confirm**.

16.2. Modify Program

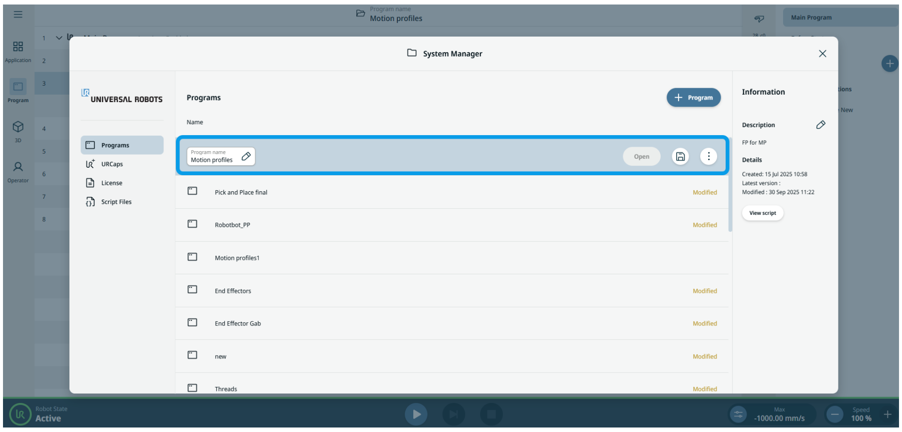
Description

The **Modify** program allows you to make changes to the programs you created for the robot.

To access Modify program

1. In the main screen, tap the **Operator** icon  on the main navigation.
2. Tap **Program name** found topmost. The **System Manager** screen appears.

   In the middle of the screen, choose and tap the program you want to modify from the list of programs you created. Once you tap the chosen program, the tab color highlights to blue.

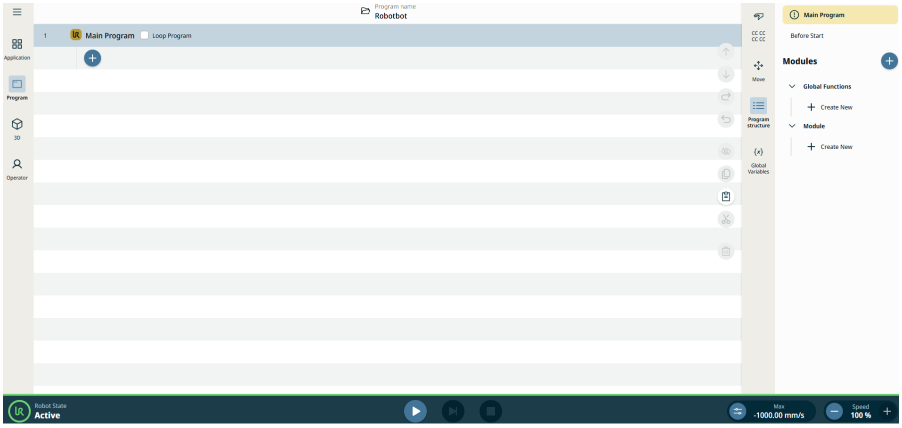
   The tab contains four buttons:

   - **Program name**
   - **Open**
   - **Save icon**
   - **Kebab icon** 

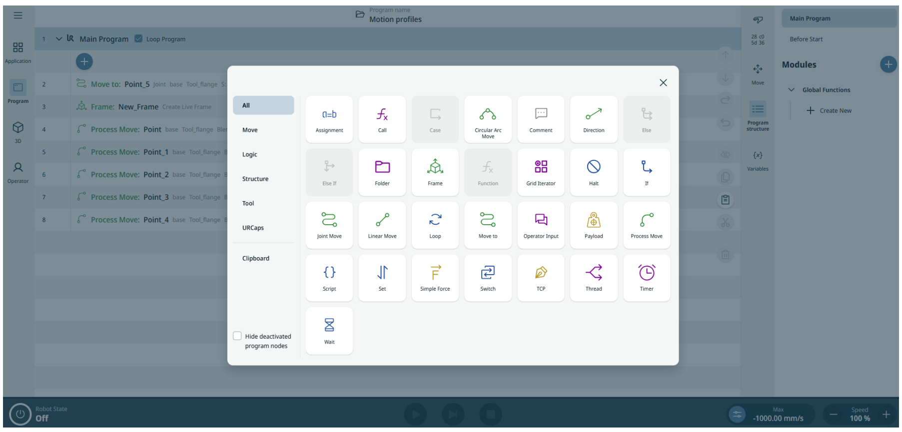
   You can see the details of the date created and the latest version of the program on the right side of the screen. A button to view the script is also provided.
3. Tap the **Program name** button to edit the name and **Confirm**.

4. Tap **Open** button. You are directed to the Main Program screen, and in the sidebar, you can access:
   - **Global Variables**
   - **Program Structure**
   - **Move**
   - **Safety Overview**

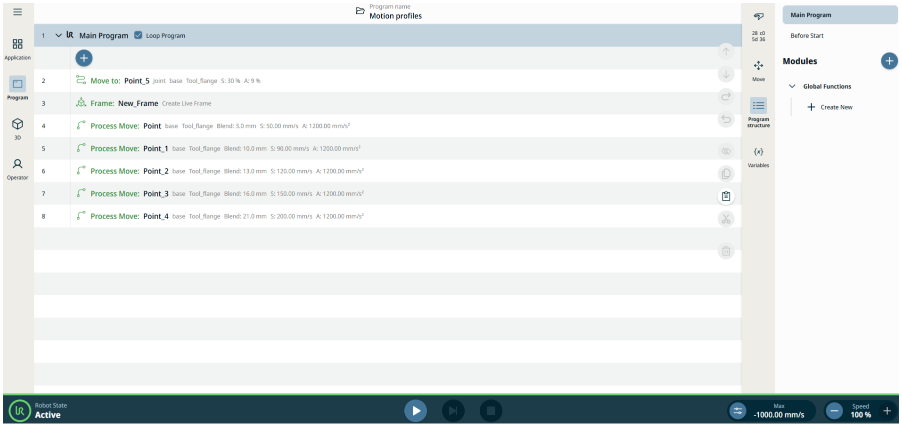
   Further to the right of the sidebar, you can access:
   - **Main Program**
   - **Before Start**
   - **Modules tree**



5. In the main program screen, tap the **add icon**. A new screen with the **Commands** and **Clipboard** toolboxes. In the Commands toolbox, you can choose which command to use to modify a program.



6. After choosing your commands to program a robot, a corresponding list is shown in the Main Program screen.



7. Tap again the **Program name** tab found at the topmost.
8. Tap the **save icon**, and you have three options:
   - **Save**
   - **Save as**
   - **Discard changes**
9. Tap the **kebab icon**, and you have three options:
   - **Duplicate**
   - **Delete**
   - **Export**

16.3. Command Nodes

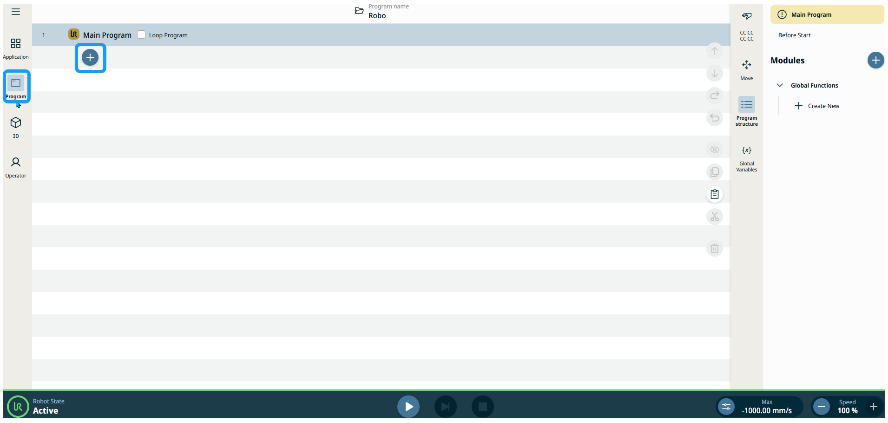
Description

The PolyScope X command nodes contain the commands that robot users apply to create a program in the simplest way.
The nodes are categorized and colored to provide the following benefits:

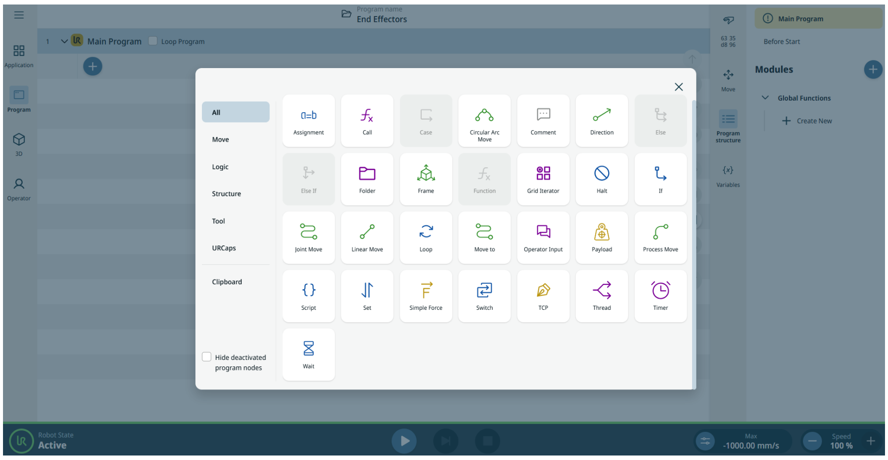
- Provide immediate identification with other nodes of similar properties.
- Highlight relationships and hierarchies in the program structure.
- Track the flow of program execution and identify issues when detecting an error.
- Indicate where the program stops without having to go through the entire program.

To access Command nodes

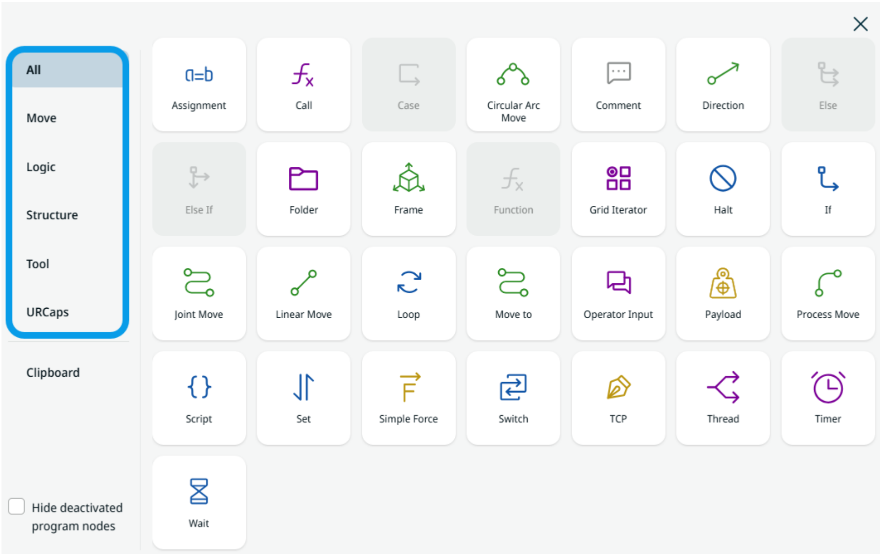
1. Tap **Program** on the main navigation, and then the **Add** icon in the main screen.



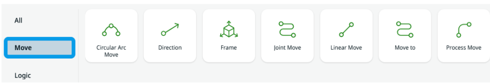
The **Commands toolbox** appears as the default screen, which contains all the command nodes.



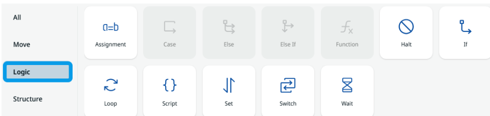
2. On the left-hand side of the screen, you have the option to view the commands by category. By default, **All** command nodes are shown.



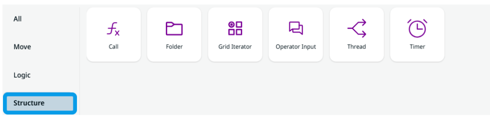
3. Tap **Move**, and the seven move command nodes are shown.



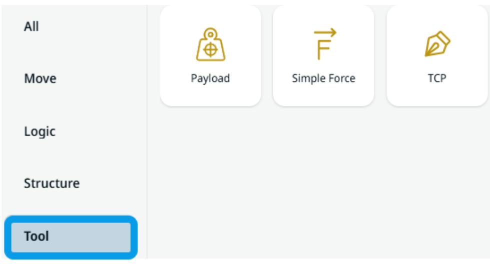
4. Tap **Logic**, and the twelve logic command nodes are shown, including the inactive ones.



5. Tap **Structure**, and the six structure command nodes are shown.

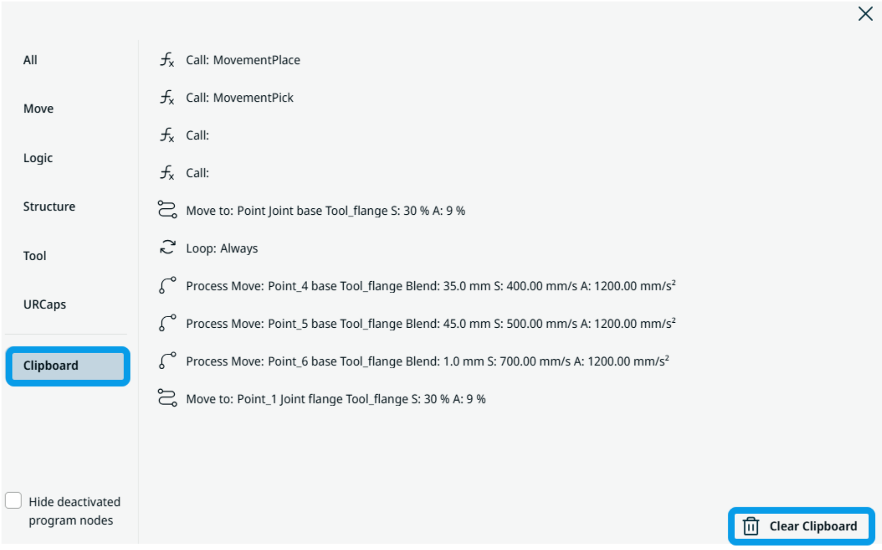


6. Tap **Tool**, and the three tool command nodes are shown.



7. When you have URCaps added in the System Manager, these become visible when you tap **URCaps** after **Tool**.

8. Tap the **Clipboard** toolbox, and you can see and review the history of cut-and-copied commands used. You also have the option to clear the clipboard at the bottom right side of the screen.



9. If you prefer to view only the active command nodes, tap **Hide deactivated program nodes**.

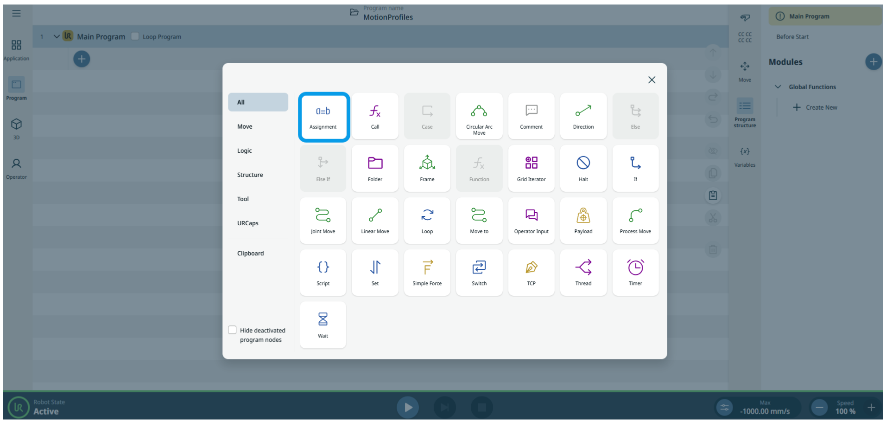
16.3.1. Assignment

Description

The **Assignment** command enables you to assign values to created variables.

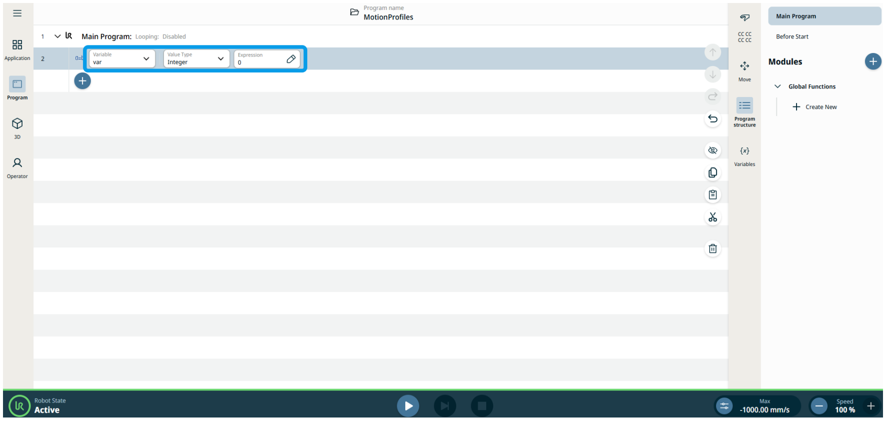
To access Assignment command

1. Go to the main program screen. See Command Nodes.
2. Tap the **Assignment** icon in the Commands toolbox.



A node with three editable fields is inserted into the program tree:

- Variable
- Value Type
- Expression



3. Tap the **Variable** field to choose your variable:
   - grid
   - grid_iterator
   - var

The Variable field contains the name of the variable. By default, the variable is called **var**.

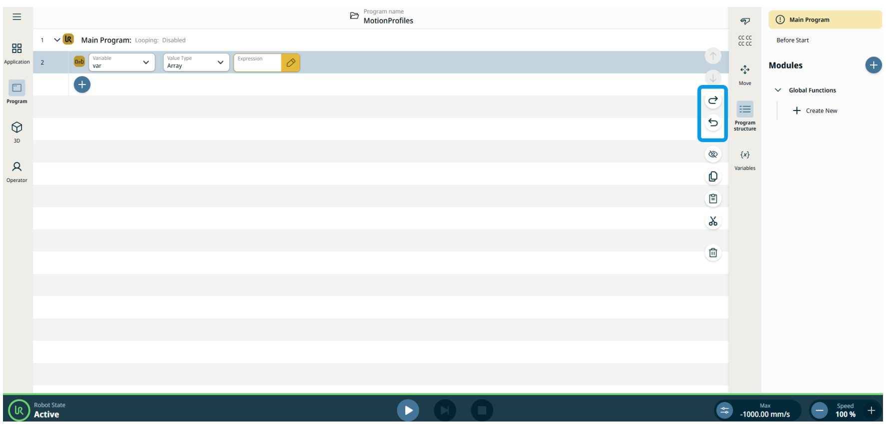
4. Tap the **Value Type** field to and select your choice:
   - Array
   - Boolean
   - Float
   - Grid
   - Integer
   - Matrix
   - Pose
   - String
   - Timer
   - Waypoint

   The Value Type contains the type of variable you are creating.

Type	Allowed Values/Examples	Examples
Array	A list of values	
Boolean	True/False	True
Float	A decimal point value	1.23
Grid	A grid defined in the application	
Integer	A whole number	5
Matrix		
Pose	A pose for the robot to move	p[0,0,0,0,0,0,]
String	Text	“Hello World”
Waypoint	A waypoint is defined in the program	

5. Tap the **Expression** field, select your variables and expression, and **Confirm**. The Expression is the value to assign the variable. See Expression Editor.

   Note: Tap the icon **arrow left** and **arrow right** in the right sidebar of the main screen if you want the assignment to be reverted.



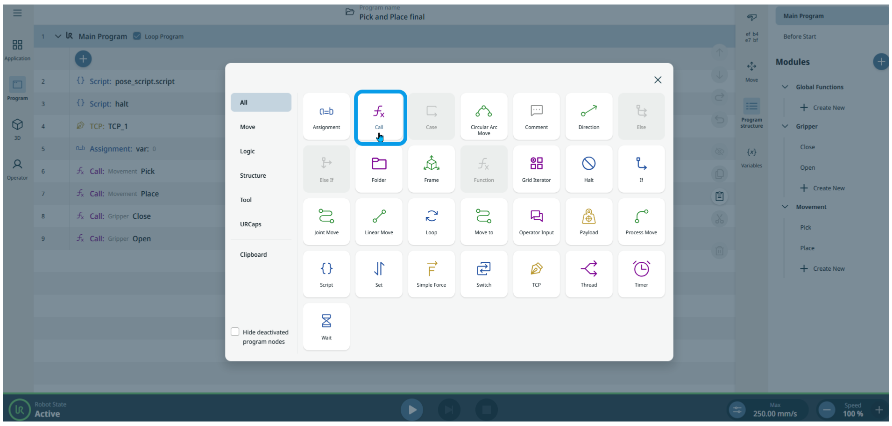
16.3.2. Call

Description

The **Call** command enables you to execute and run created global functions and modules, which are seen in the multitask screen. To use this command, a prior program creation using modules and functions is a requisite. See Modules and Functions on how to set up commands and group them into modules and functions. Read Gripper Module and Movement Module for details about modules. Read Open/Close and Pick/Place for details about functions.

To access Call command

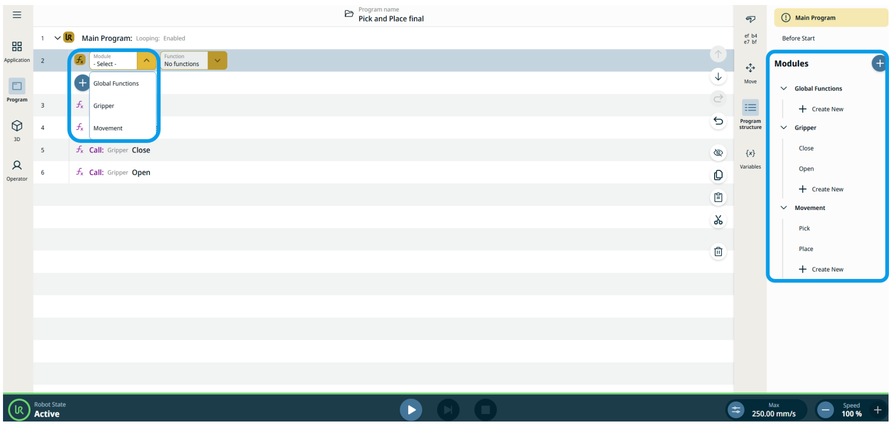
1. Open an existing program where commands are grouped in modules and functions. In this example, open the Pick and Place program.
2. Go to the main program screen. See Command Nodes.
3. Tap the **Call** icon in the Commands toolbox.



An editable **Module** field is inserted into the program tree.

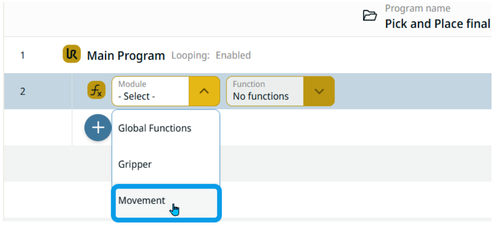
4. Tap the **Module** field.

In addition to **Global Functions**, you can see the created modules: **Gripper** and **Movement**. The list of modules correspond to the ones in the Modules tree in the multitask screen.



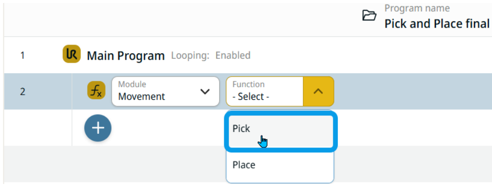
Call Movement module

5. Select **Movement** in the Module field.



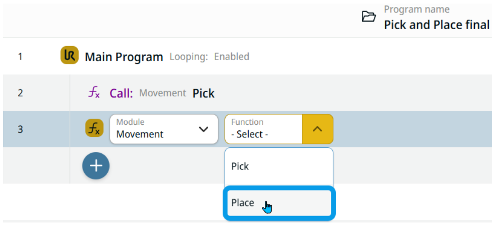
Call a Pick function

6. After the module is selected, the **Function** field is activated. Select **Pick** function.



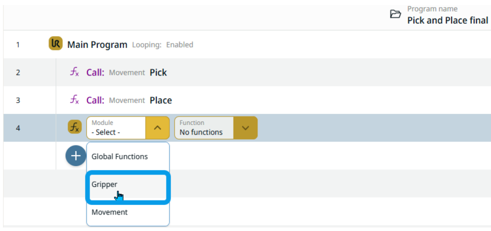
Call a Place function

7. Repeat steps 2-5.
8. In the same **Function** field, select **Place** function.

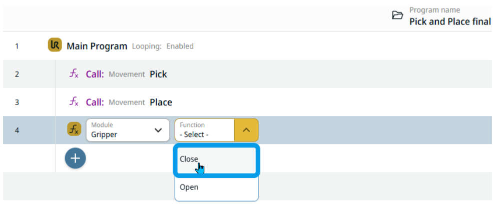


Call Gripper module and Close function

9. Tap the add icon and then the Call icon.
10. Select **Gripper** in the Module field.

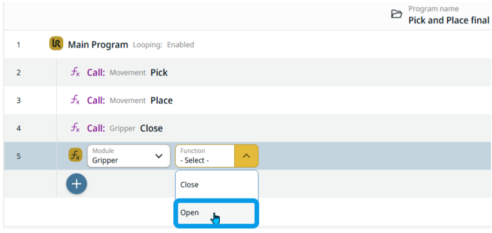


11. In the **Function** field, select **Close** function.



Call an Open function

7. Repeat steps 2-5.
8. In the same **Function** field, select **Open** function.



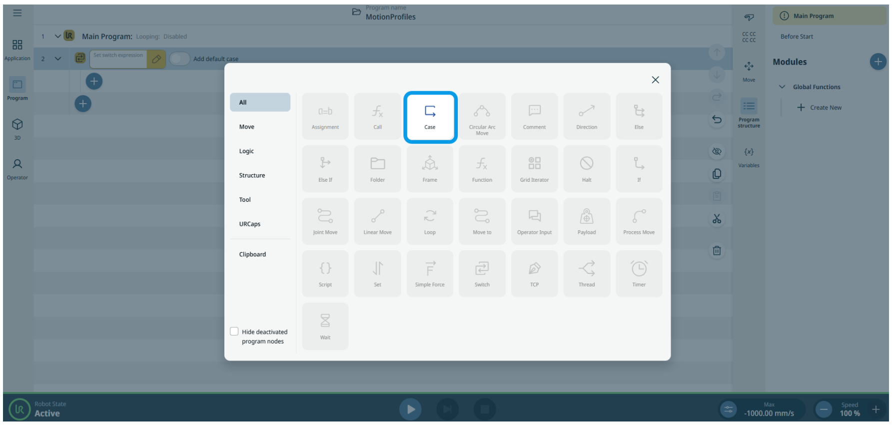
16.3.3. Case

Description

The **Case** command enables you to point to a function to be executed, depending on an input. It can only be used with the Switch command. It is usually referred to as "Switch-Case" as they always go together.

To access Case command

1. Go to the main program screen. See Command Nodes.
2. Access the Switch command.
3. After the input in **Set switch expression**, tap the first add icon from the child node of the switch command.
4. Tap **Case** icon in the Commands toolbox.

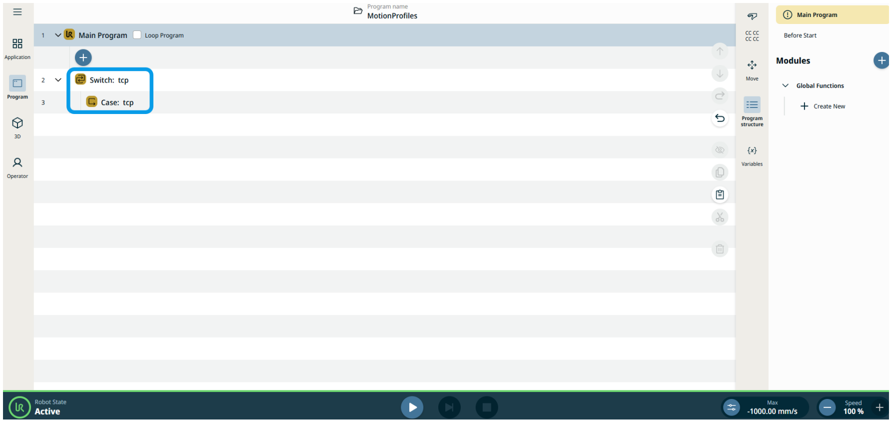


   A node with the **Set value** field and **Module** field is inserted into the program tree, under the switch expression.

5. Tap **Set value**.
6. Create the set value and tap **Confirm**.
7. Tap **Module**.

8. Tap **Global Functions**.

   A switch-case is created in the program tree.



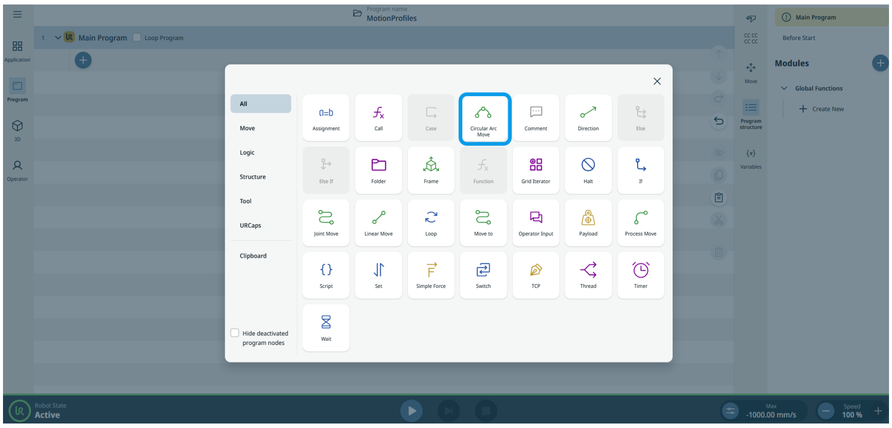
16.3.4. Circular Arc Move

Description

The **Circular Arc Move** enables you to program the robot to move in a circular motion while keeping the tool orientation either fixed or unconstrained.

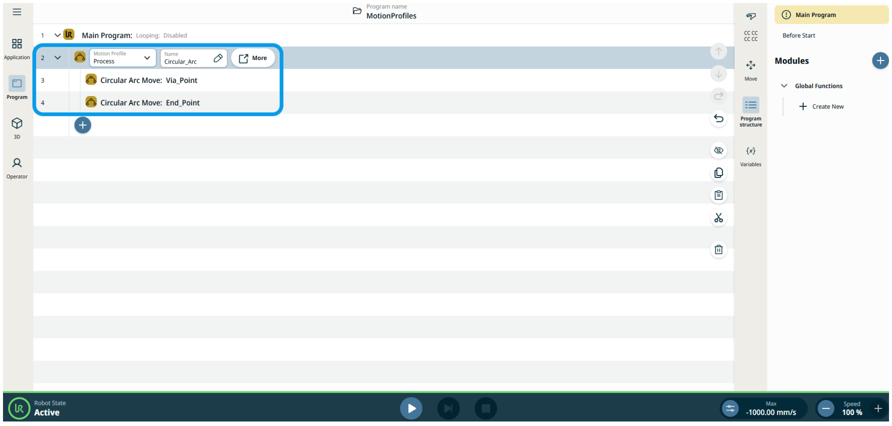
To access Circular Arc Move command

1. Go to the main program screen. See Command Nodes.
2. Tap the **Circular Arc Move** icon in the Commands toolbox.



A parent node and two child nodes are inserted into the program tree.

- First child node = Circular Arc Move: Via_Point
- Second child node = Circular Arc Move: End_point



The node has three editable fields and a **More** button for advanced options.

The three editable fields are:

- **Name**
  A name for referencing the node.
- **Speed**
  The robot speed during the move.
- **Acceleration**
  The robot acceleration during the move.

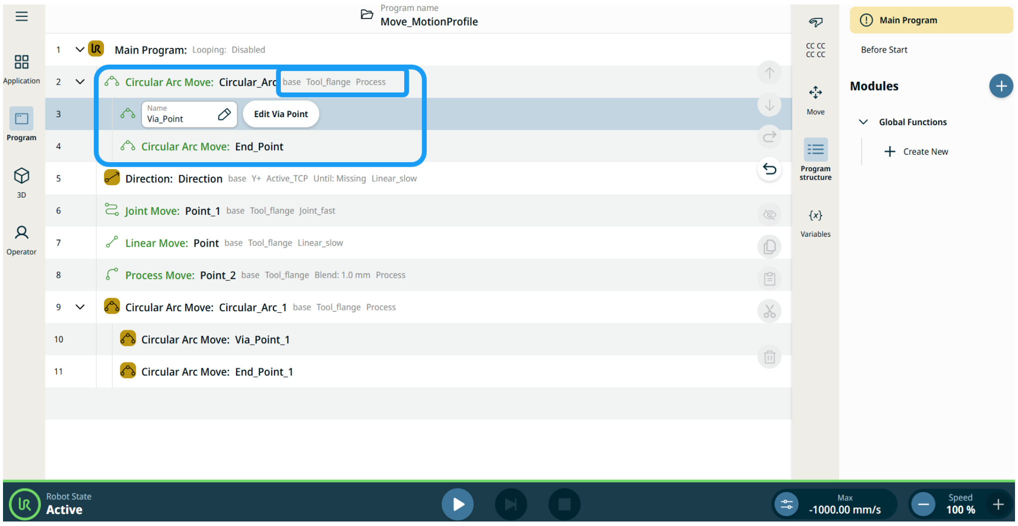
3. Tap the **Name** field, and three tabs appear:
   - Value
   - Variable
   - Expression
4. Enter the data in each tab and tap **Confirm**. See Value-Variable-Expression Tabs.
5. Tap the **Speed** field, and three tabs appear:
   - Value
   - Variable
   - Expression
6. Repeat step 4.
7. Tap the **Acceleration** field, and three tabs appear:
   - Value
   - Variable
   - Expression
8. Repeat step 4.

Motion Profile labels on Move Nodes

All **Move-related command nodes** in the program tree display the **Motion Profile** associated with each node. This feature enables users to see the selected Motion Profile on a Move node. This feature is implemented on the following command nodes:

- Linear Move
- Joint Move
- Process Move
- Direction
- 16.3.4 Circular Arc Move on page 126

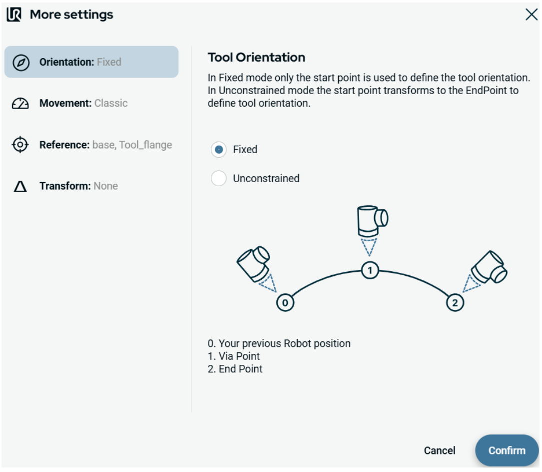
> **NOTICE**
> Move nodes with Custom motion profile will display the values for the speed and acceleration instead of Motion profile.


Program name
Move_MotionProfile
Application
Program
3D
Operator
1 Main Program: Looping: Disabled
2 Circular Arc Move: Circular_Arc base Tool_flange Process
3 Name Via_Point Edit Via Point
4 Circular Arc Move: End_Point
5 Direction: Direction base Y+ Active_TCP Until: Missing Linear_slow
6 Joint Move: Point_1 base Tool_flange Joint_fast
7 Linear Move: Point base Tool_flange Linear_slow
8 Process Move: Point_2 base Tool_flange Blend: 1.0 mm Process
9 Circular Arc Move: Circular_Arc_1 base Tool_flange Process
10 Circular Arc Move: Via_Point_1
11 Circular Arc Move: End_Point_1
Move
Program structure
Variables
Main Program
Before Start
Modules
Global Functions
Create New
Robot State
Active
Max
-1000.00 mm/s
Speed
100 %

To access More settings

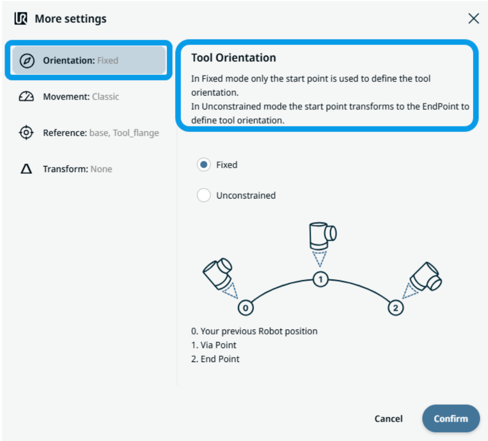
Tap the **More** button, and the **More settings** screen appears.
On the left side, you see four advanced options:

- Orientation
- Movement
- Reference
- Transform



To use Orientation

1. Tap **Orientation**.



   On the right side, two buttons are provided:

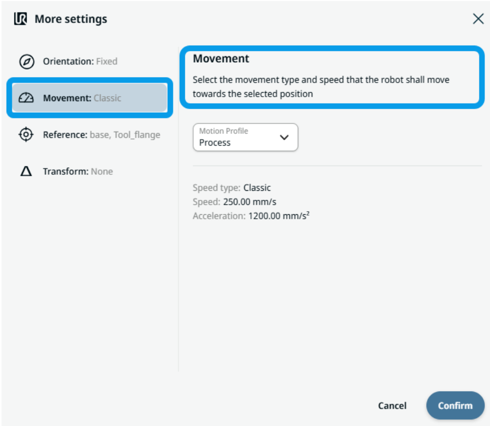
   - Fixed
   - Unconstrained

2. Tap **Fixed** when you want the start point only to be used to define the tool orientation, and **Confirm**.
3. Tap **Unconstrained** when you want the start point to transform to the endpoint to define tool orientation and **Confirm**.

   Note: A diagram shows the difference between the two endpoints when you choose unconstrained.

To use Movement

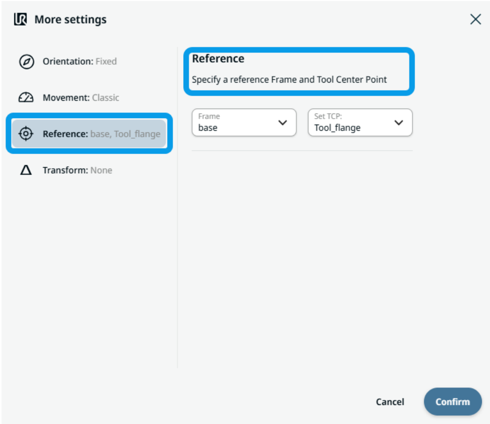
1. Tap **Movement**.



   Two fields appear:
   - Speed
   - Acceleration
2. Tap the **Speed** field, and three tabs appear:
   - Value
   - Variable
   - Expression
3. Enter the data in each tab and tap **Confirm**. See Value-Variable-Expression Tabs.
4. Tap the **Acceleration** field, and three tabs appear:
   - Value
   - Variable
   - Expression
5. Enter the data in each tab and tap **Confirm**. See Value-Variable-Expression Tabs.

To use Reference

1. Tap **Reference**, and two editable **Frame** and **TCP** fields appear.

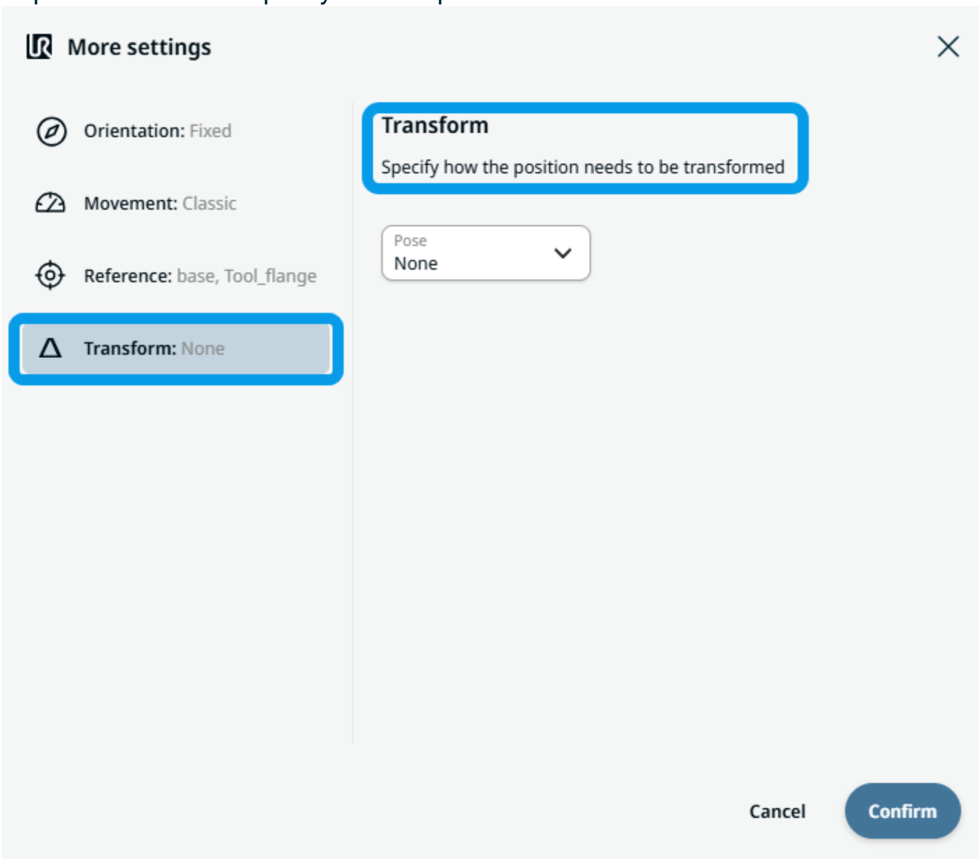


   Tap the **Frame** field, and four options are selectable:
   - world
   - base
   - flange
   - tcp
2. Tap the **TCP** field, and two options are selectable:
   - Tool_flange
   - Active_TCP
3. In the **Position** column, tap **X, Y, Z** fields to enter the value, and **Confirm**.
4. In the **Rotation** column, tap **RX, RY, RZ** fields to enter the value, and tap **Confirm**.

To use Transform

1. Tap **Transform**.

   Tap the **Pose** field to specify how the position needs to be transformed and **Confirm**.

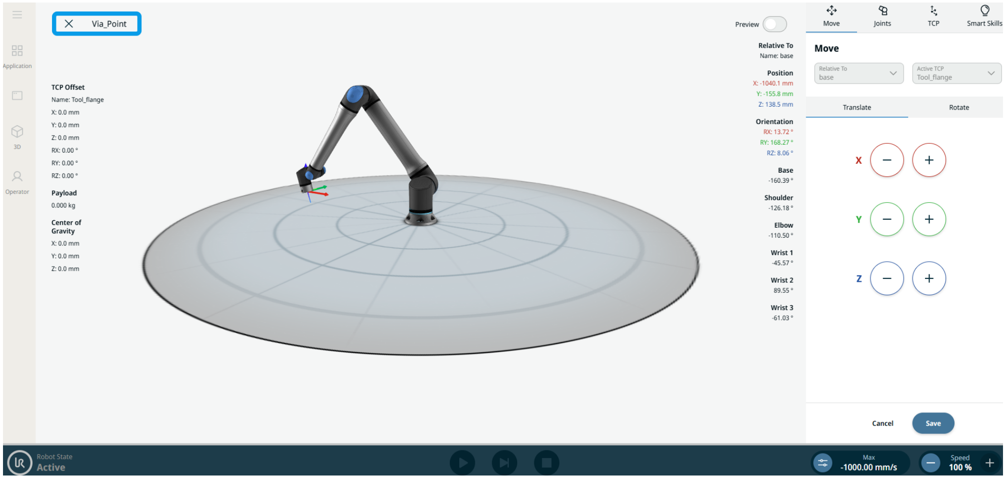


2.

To use Circular Arc Move: Via_ point

1. Tap the **Circular Arc Move: Via_Point** node and two fields appear:
   - Name
   - Teach Via Point
2. Tap the **Name** field, and three tabs appear:
   - Value
   - Variable
   - Expression
3. Enter the data in each tab and tap **Confirm**. See Value-Variable-Expression Tabs.
4. Tap **Teach Via Point**.

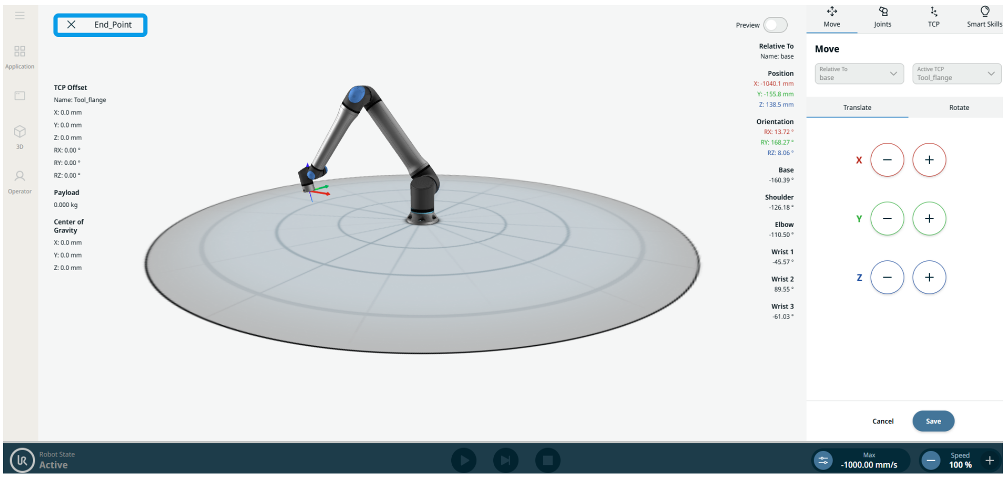
   The 3D viewer screen of the robot appears. (See Joint Move and 16.3.16 Linear Move on page 163 for details of use.)



To use Circular Arc Move: End_Point

1. Tap the **Circular Arc Move: End_Point** node, and three fields appear:
   - Name
   - Blend
   - Teach Via Point
2. Tap the **Name** field, and three tabs appear:
   - Value
   - Variable
   - Expression
3. Enter the data in each tab and tap **Confirm**. See Value-Variable-Expression Tabs.
4. Tap the **Blend** field, and three tabs appear:
   - Value
   - Variable
   - Expression
5. Repeat step 3.
6. Tap **Teach End Point**.

   The 3D viewer screen of the robot appears. (See Joint Move and 16.3.16 Linear Move on page 163 for details of use.)



16.3.5. Comment

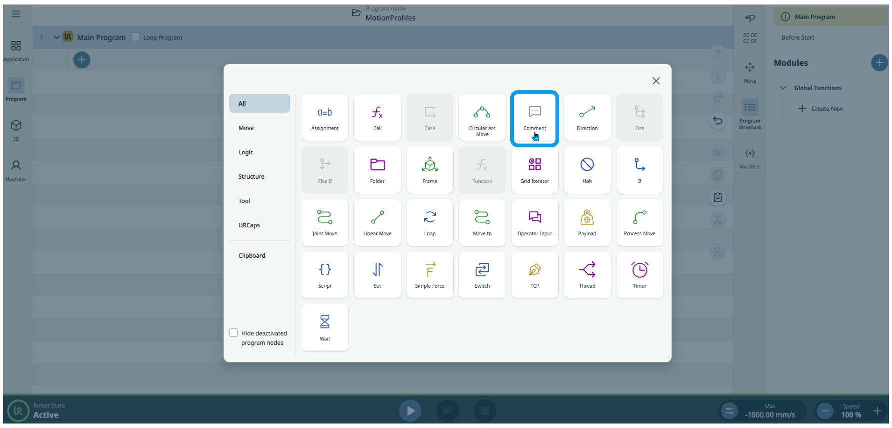
Description

The **Comment** command allows you to add text descriptions into your program and so enables allows you to keep track of decisions you make while you create or update your robot program.

To access Comment command

1. Go to the main program screen. See Command Nodes.
2. Tap the **Comment** icon in the Commands toolbox.

   An editable **Comment** field is inserted into the program tree.



3. Tap the Comment field.
4. Create your comment.

   Note: A maximum of 1,000 characters is allowed.
5. Tap **Confirm**.

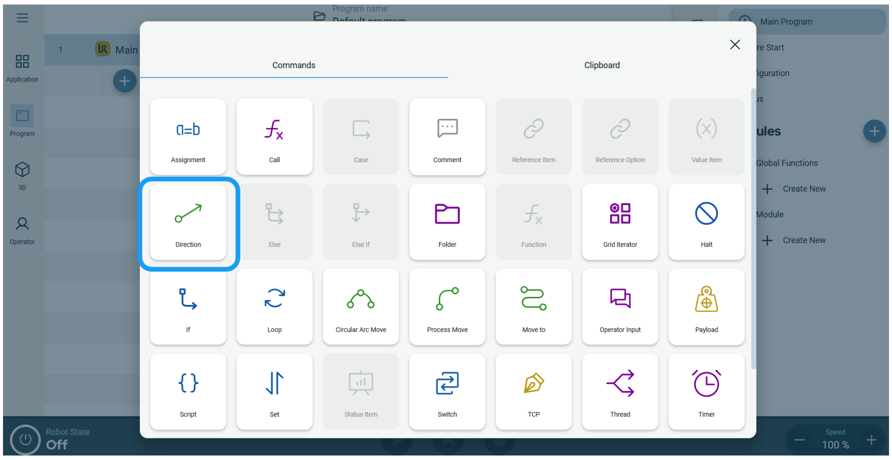
16.3.6. Direction

Description

The **Direction** command enables you to move the robot in a specific direction in a linear motion along an axis. A stop condition halts it.

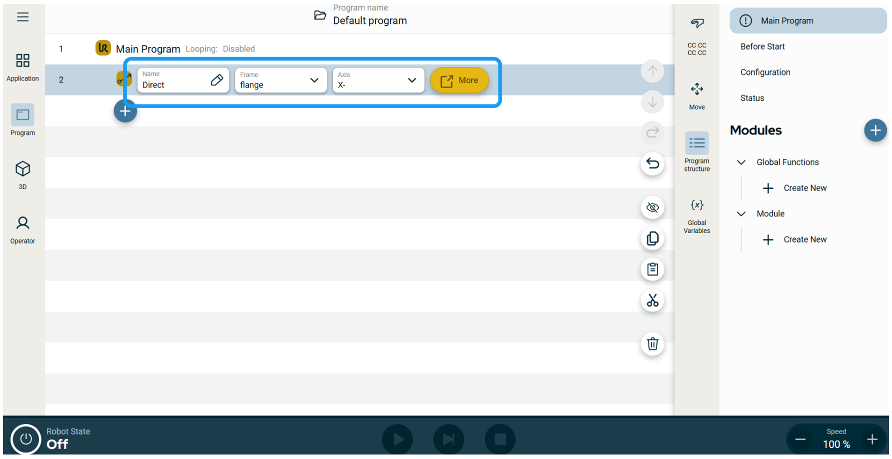
To access Direction command

1. Go to the main program screen. See Command Nodes.
2. Tap the **Direction** icon in the Commands toolbox.



A node with three editable fields and a **More** button is inserted into the program tree. The three fields are:

- Name
- Frame
- Axis



3. Tap the **Name** field.
4. Enter the Value name in the tab and tap **Confirm**.

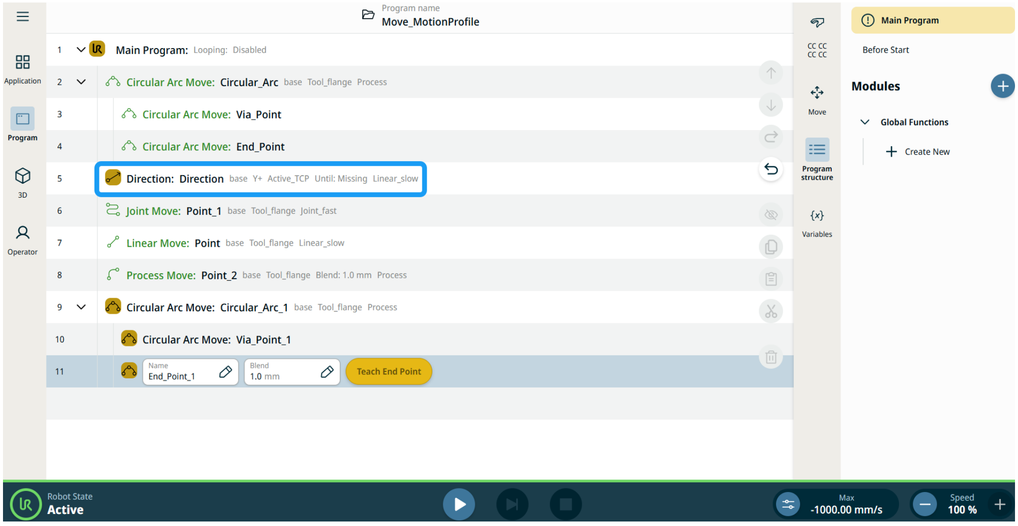
5. Tap the **Frame** field, and four choices are provided:
   - world
   - base
   - flange
   - tcp
6. Select your preferred frame choice.
7. Tap the **Axis** field, and the six axes are provided:
   - X+
   - X-
   - Z+
   - Z-
   - Y+
   - Y-
8. Tap your chosen axis.

Motion Profile labels on Move Nodes

All **Move-related command nodes** in the program tree display the **Motion Profile** associated with each node. This feature enables users to see the selected Motion Profile on a Move node. This feature is implemented on the following command nodes:

- Linear Move
- Joint Move
- Process Move
- Direction
- 16.3.4 Circular Arc Move on page 126

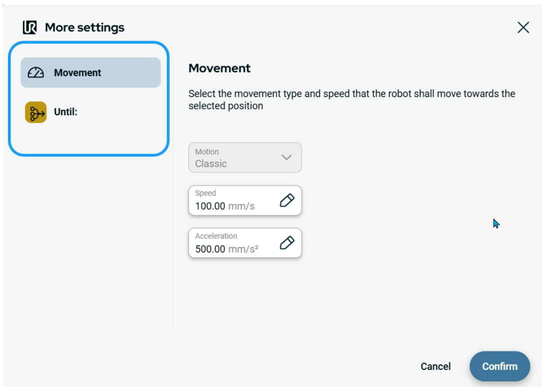
> **NOTICE**
> Move nodes with Custom motion profile will display the values for the speed and acceleration instead of Motion profile.



To use the More options

Tap the **More** button.
A new screen appears containing two advanced settings:

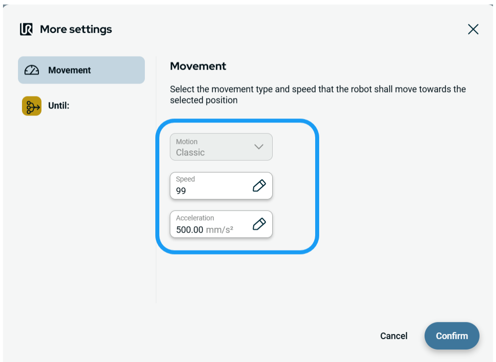
- Movement
- Until



To use Movement setting

1. On the left side, tap **Movement**, and three fields appear:
   - Motion
   - Speed
   - Acceleration

   Note: The Motion field is inactive.

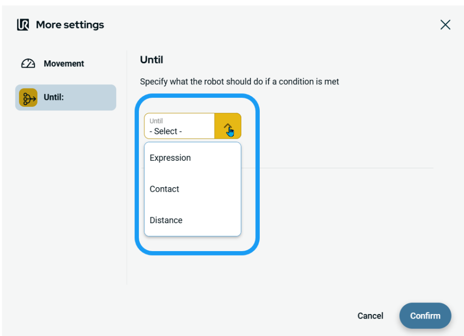
2. Tap the **Speed** field, and three tabs are available:
   - Value
   - Variable
   - Expression

   Note: The Variable tab is inactive.
3. Enter the data in each tab and tap **Confirm**. See Value-Variable-Expression Tabs.
4. Tap the **Acceleration** field, and three tabs are available:
   - Value
   - Variable
   - Expression

   Note: the Variable tab is inactive.
5. Enter the data in each tab and tap **Confirm**. See Value-Variable-Expression Tabs.

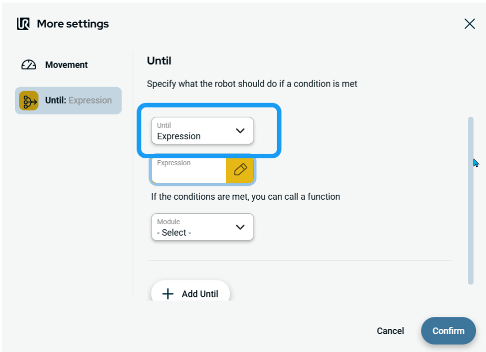
To use Until setting

1. On the left side, tap **Until**. An **Until** field appears, giving you three conditions to add:
   - Expression
   - Contact
   - Distance

To access Expression

1. Tap **Expression**, and three functions are added:
   - Expression field
   - Module Select drop-down field
   - Add until button

   Expression. Sets the expression to check against, and you can optionally call a function when the condition is met.
2. Tap the **Expression** field, enter your preferred expression, and **Confirm**.
3. Tap the **Module** field, and two selections are available:
   - Global Functions
   - Module
4. Select either **Global Functions** or **Module**.

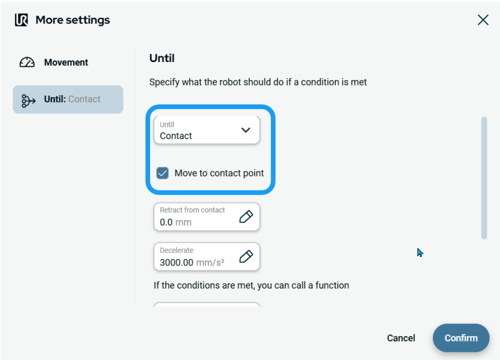
   An additional **Function** field appears to the right.
5. Tap **Add Until** button to add further conditions, and **Confirm**.

   You can delete an Until condition by clicking the thrash can, but the last condition cannot be deleted.

To access Contact

1. Tap **Contact**, and five functions are added:
   - Checkbox for **Move to contact point**
   - **Retract from contact** field
   - **Decelerate** field
   - **Module Select** field
   - **Add Until** button

   Until contact checks for tool contact.

2. Check the **Move to contact point** box.
3. Tap the **Retract from contact** field, and three tabs are available:
   - Value
   - Variable
   - Expression

   Note: The Variable tab is inactive.
4. Enter the data in each tab and tap **Confirm**. See Value-Variable-Expression Tabs.

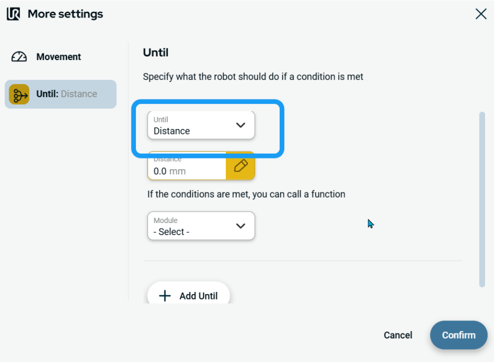
   Note: When you uncheck the **Move to contact point** box, the **Retract from contact** field becomes inactive.
5. Tap the **Decelerate** field, and three tabs are available:
   - Value
   - Variable
   - Expression

   Note: The Variable tab is inactive.
6. Enter the data in each tab and tap **Confirm**. See Value-Variable-Expression Tabs.
7. Tap the **Module Select** field, and two selections are available:
   - Global Functions
   - Module
8. Tap either **Global Functions** or **Module**.

   An additional **Function** field appears to the right.
9. Tap **Add Until** button to add further conditions, and **Confirm**.

To access Distance

1. Tap **Distance**, and four functions are added:
   - **Distance** field
   - **Module** field
   - **Function Select** field
   - **Add Until** button



2. Tap the **Distance** field, and three tabs are available:
   - Value
   - Variable
   - Expression

   Note: The Variable tab is inactive.
3. Enter the data in each tab and tap **Confirm**. See Value-Variable-Expression Tabs.
4. Tap either **Global Functions** or **Module**.

   An additional **Function** field appears to the right.
5. Tap **Add Until** button if there is a need for you to use further Until function.
6. Tap **Add Until** button to add further conditions, and **Confirm**.

16.3.7. Else

Description

The **Else** command, in addition to Else If, is used together with the If command. It enables you to make statements to change the robot's behavior based on sensor inputs or variable values.
It is only insertable directly after an If or Else If node.
The children of this node are only executed if the preceding If or Else If is false.

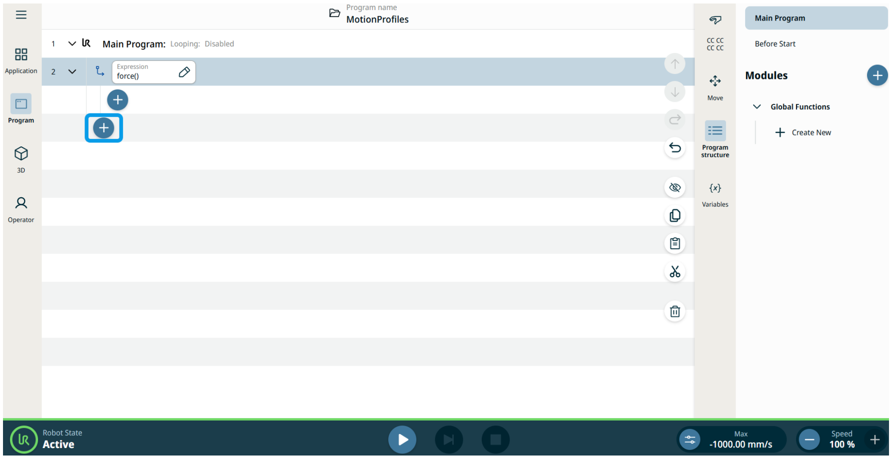
To access Else command

1. Go to the main program screen. See Command Nodes.
2. Tap the **If** icon in the Commands toolbox.

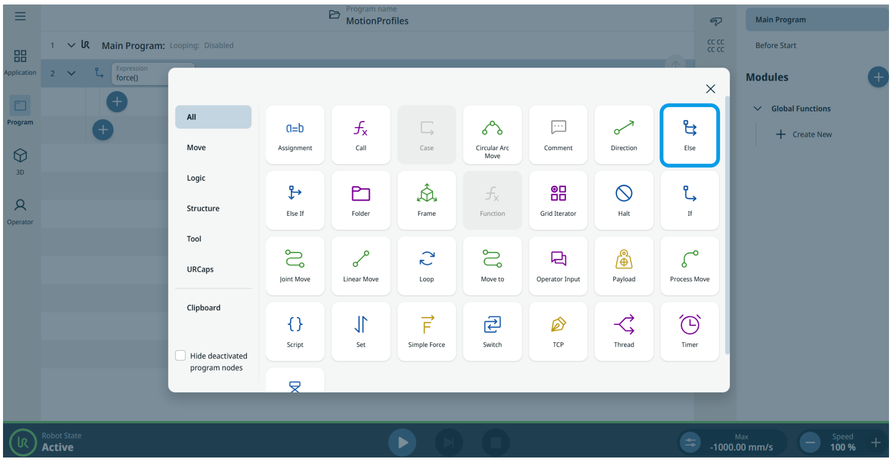
   Follow the process when accessing the If program. See If for more.

   An Expression field and two add icons are inserted into the program tree:

   - add icon in the If child node
   - add icon for another node
3. Tap the **second add icon**.

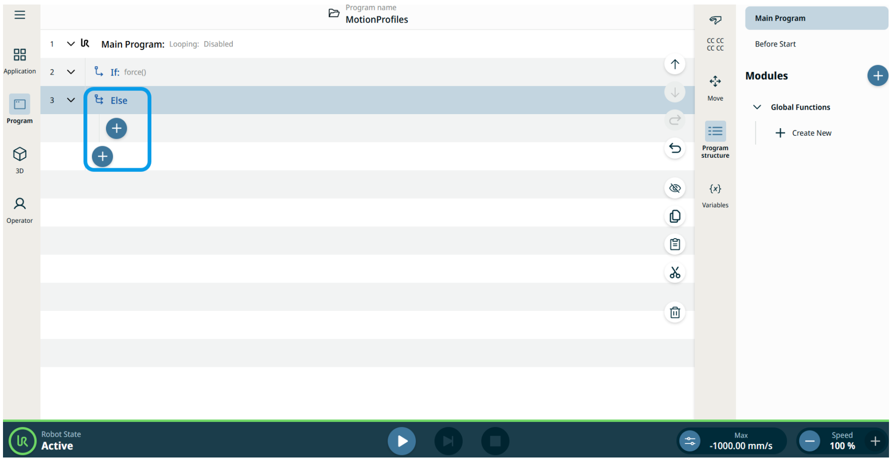


   The Commands toolbox appears.
4. Tap the **Else** icon in the Commands toobox.



The Else node is inserted into the program tree.

5. Choose and tap the appropriate add icon to access other commands for the robot movement.



16.3.8. Else If

Description

The **Else If** command, in addition to Else, is used together with the If command. It enables you to make statements to change the robot's behavior based on sensor inputs or variable values.
The Else If node is only insertable directly after an If or Else node.
The children of this node are only executed if the preceding If or Else If node is False, but its expression evaluates it as True.

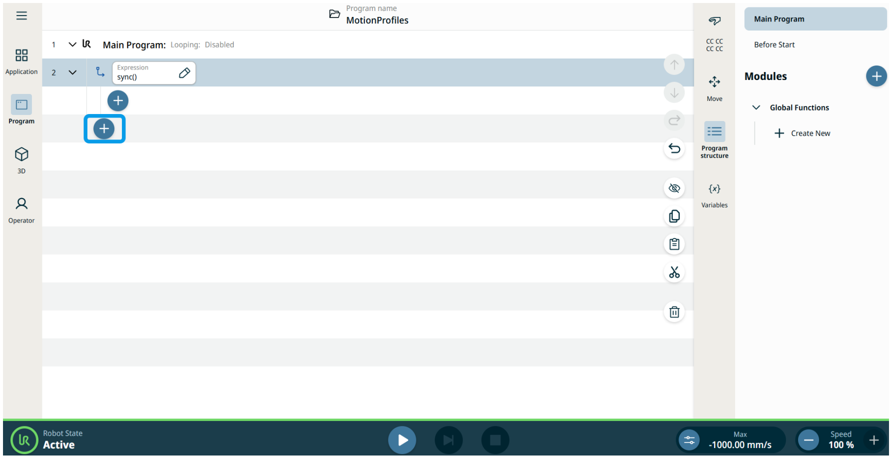
To access Else If command

1. Go to the main program screen. See Command Nodes.
2. Tap the **If** icon in the Commands toolbox.

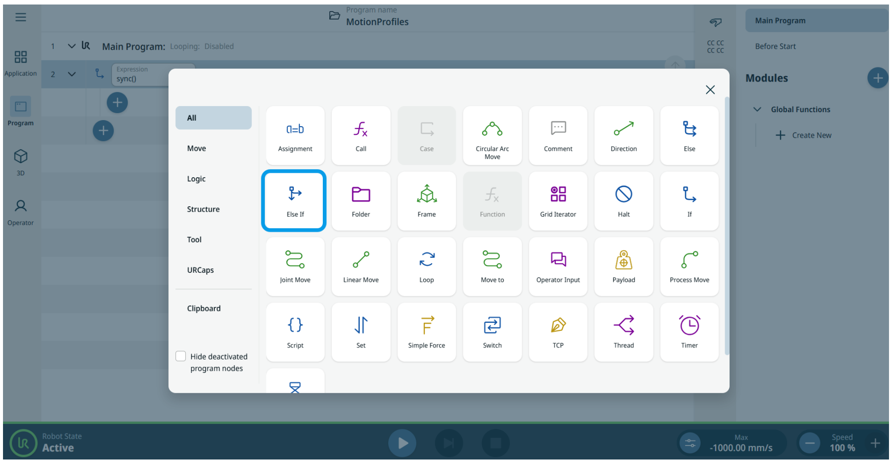
   Follow the process when accessing the If program. See If for more.

   An **Expression** field and two add icons are inserted into the program tree:

   - Add icon in the If child node
   - Add icon for another node
3. Tap the **second add icon**.

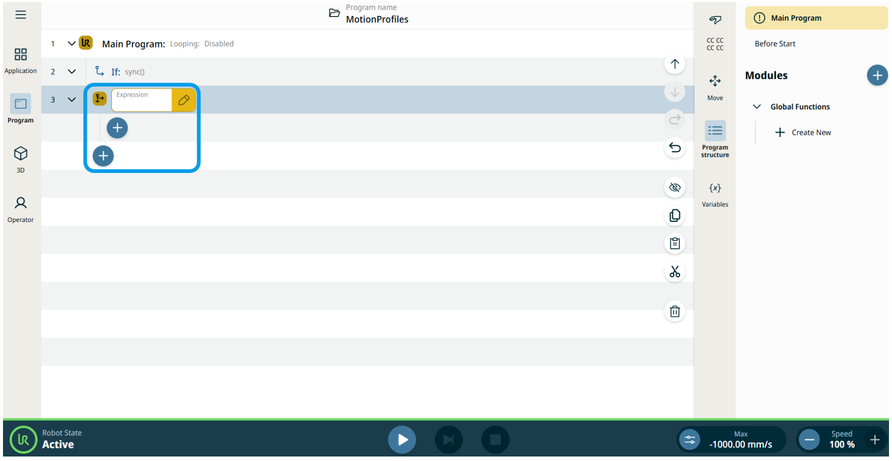


   The Commands toolbox appear.
4. Tap the **Else If** icon in the Commands toolbox.



5. The Else If node is inserted into the program tree.

   The node has an editable **Expression** field and two add icons.



6. Tap the **Expression** field.
7. Create your expression .
8. Tap **Confirm**.
9. Choose and tap the appropriate **add icon** to access other commands for the robot movement.

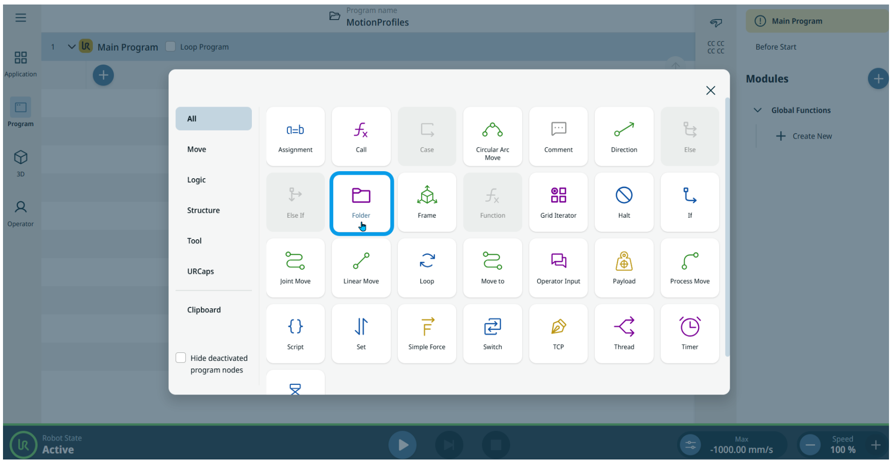
16.3.9. Folder

Description

The **Folder** command allows you group nodes together in a folder to create an easy-to-read overview of your robot program.

To access Folder command

1. Go to the main program screen. See Command Nodes.
2. Tap the **Folder** icon in the Commands toolbox.



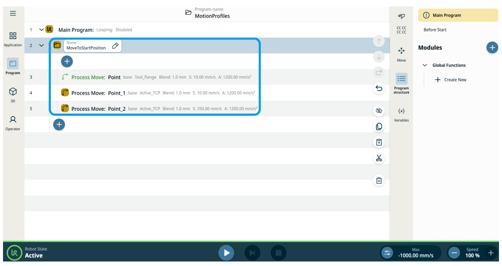
A Folder node, which is the **Name** field, is inserted into the program tree.

3. Tap the **Name** field.
4. Edit the folder name and tap **Confirm**.
5. You can see two add icons below the folder name:
   - You can add a new folder under the chosen program.
   - You can also add a subfolder to your created folder.

Example

Create Folder of all Move Commands

1. Tap the **Folder** icon.
2. Tap the **Name** field.
3. Edit folder name to **MoveToStartPosition**.
4. Tap the **first add icon** below the folder node.
5. Add the move-related commands in the program.



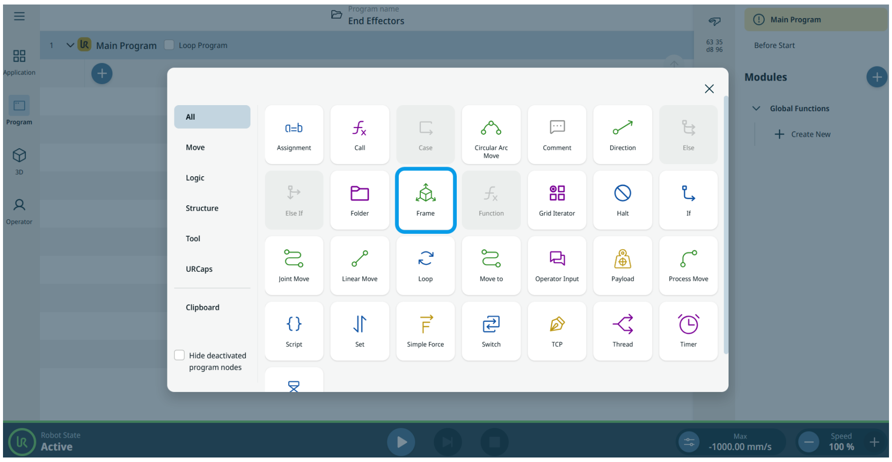
16.3.10. Frame

Description

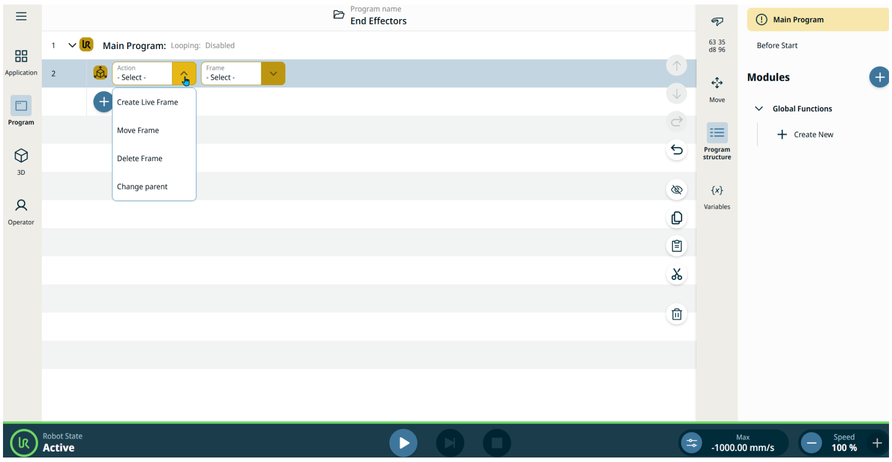
The **Frame** command node allows for the easy creation and modification of frames during the program execution. Frames created with this node are referred to as Live Frames to distinguish them from the frames created from the application section.

To access Frame command

1. Go to the main program screen. See Command Nodes.
2. Tap the **Frame** icon in the Commands toolbox. A node with two editable fields is inserted into the program tree, Action and Frame.



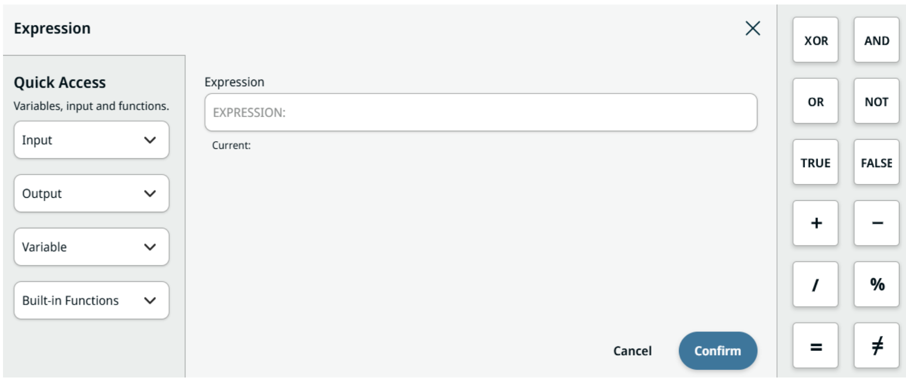
3. Tap the **Action** field, and four choices are provided:
   - Create Live Frame
   - Move Frame
   - Delete Frame
   - Change Parent



Create Live Frame

This action creates a new frame with the given name. The position of the frame can be specified from the position dialog, as well as the parent of the frame. The position can be given by either an X, Y, Z, RX, RY, RZ or by an expression.

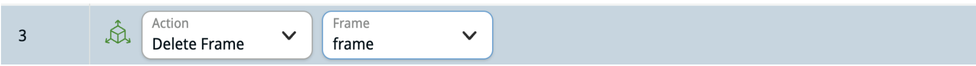
1. Create your preferred frame name in the **Name** field.
2. Tap the **Position** tab, and further field and tabs are provided to choose from.
3. The **Reference** field enables you to select world, base, flange, and TCP.
4. In the default **Pose toolbox**. you enter the **Position** in X, Y, Z and **Rotation** in RX, RY, RZ.
5. In the **Expression toolbox** , tap the **Expression field**, and related functions are provided such as the Quick Access panel of IO, Variable, and Built-in Functions.



Move Frame

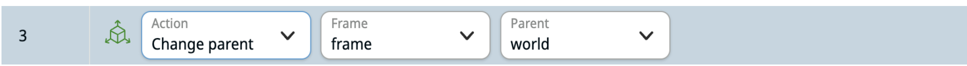
This action moves an existing frame to a new position. The frame can be either a frame created in the application or a live frame. As with the creating live frame, the position can be specified in either coordinates or an expression.
When you select Position tab, follow the instructions 2-5 provided in “Create Live Frame.”

Delete Frame



This action deletes an existing frame. The frame can be one created in the application or a live frame. Predefined frames such as base and world cannot be deleted.

Change Parent



This actions changes the parent of an existing frame to a new one.

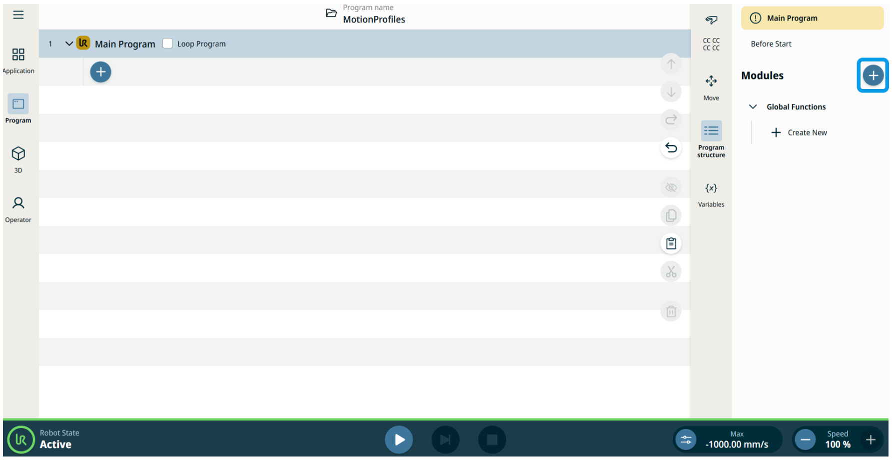
16.3.11. Function

Description

The **Function** command allows you to perform a specialized operation, like performing calculation, moving the robot, and other similar movements. The function will not be executed until the call command. It is only insertable inside a module or from the Global Functions.

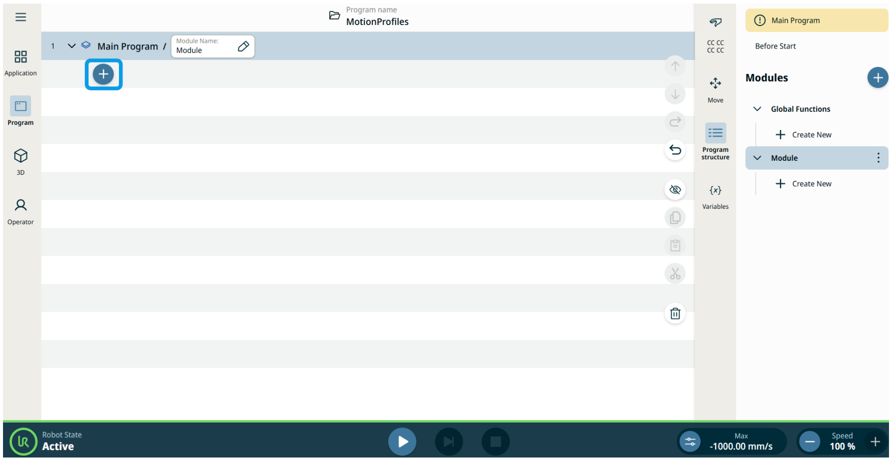
To access Function command

1. Go to the main program screen. See Command Nodes.
2. Tap the **add icon** beside the **Modules** tree in the multitask screen.

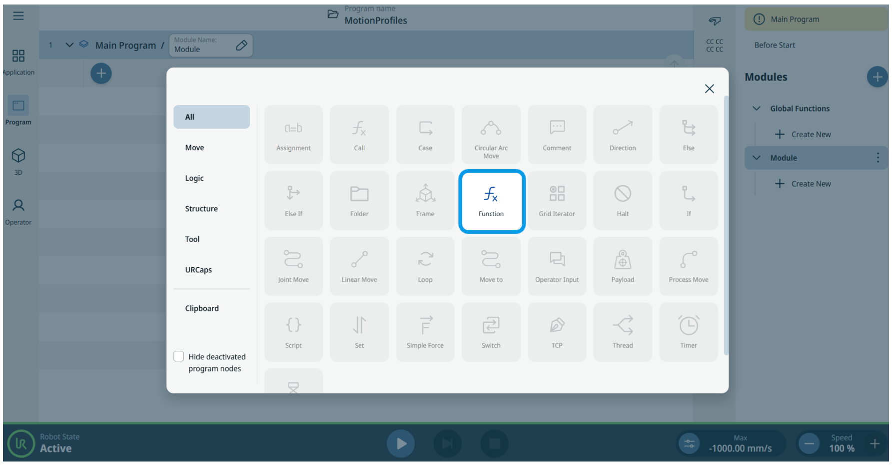


A new module is inserted into the module tree.

3. Tap the **add icon** in the main screen located under the Main Program.



4. Tap the **Function** icon in the Commands toolbox.



A node is inserted into the program tree, which is an editable field called **Function Name**.

5. Tap the **Function Name** field.
6. Create a function name.
7. Tap **Confirm**.

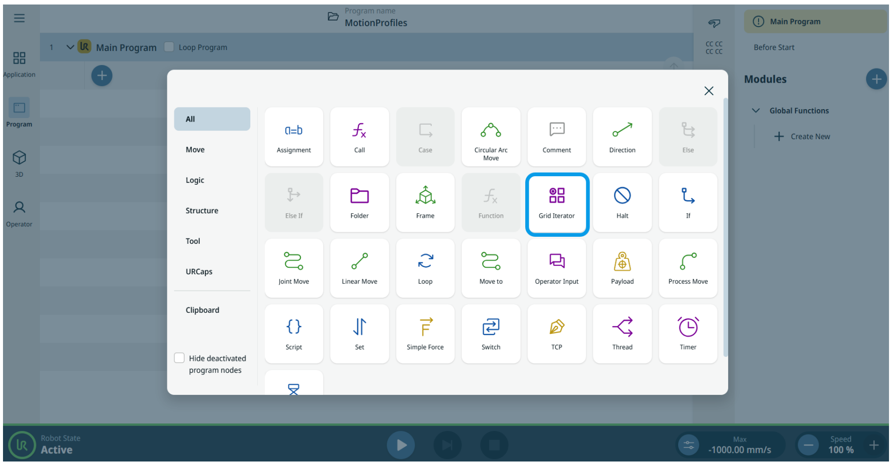
16.3.12. Grid Iterator

Description

The **Grid Iterator** command enables you to move a grid from the first position the next to first position. The **Grid Iterator** command is used in combination with a grid created in the Grids submenu under the Application menu, located on the left toolbar. See Grids Application.

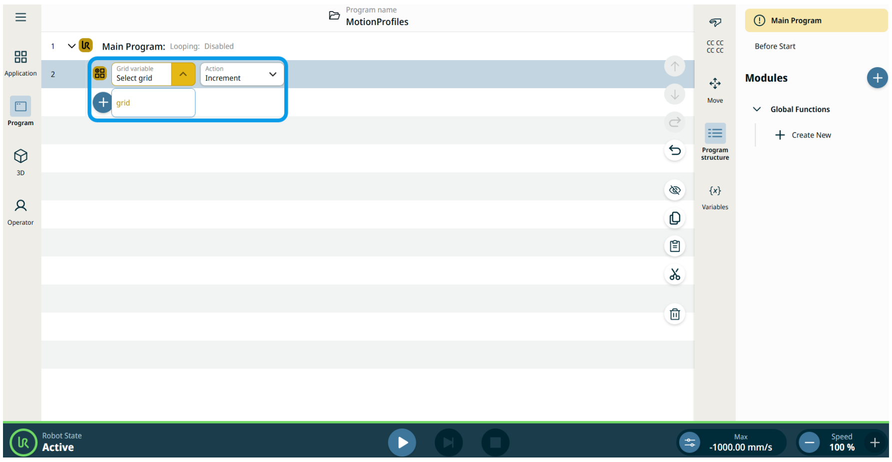
To access Grid Iterator command

1. Go to the main program screen. See Command Nodes.
2. Tap the **Grid Iterator** icon in the Commands toolbox.



A node is inserted into the program tree. The node has an editable field named **Grid variable** and **Action**.

3. Tap the **Grid variable** field.
4. Choose the particular grid you created in the **Grids submenu** of Application.



5. Tap the **Action** field.
6. Select either **Increment** or **Reset**.

Each increment moves the variable to the next grid position. The pose variable, **grid_ iterator**, can be used as the target in a Move node instead of a fixed waypoint, enabling sequential motion through the grid layout.

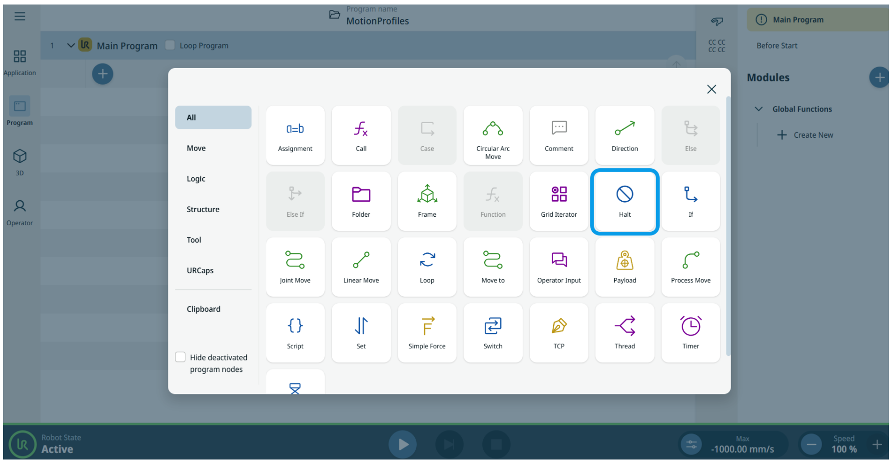
16.3.13. Halt

Description

The **Halt** command allows you to stop the robot at that location.

To access the Halt command

1. Go to the main program screen. See Command Nodes.
2. Tap the **Halt** icon in the Commands toolbox.



You see a notification under the chosen program: "Program execution stops at this point."

16.3.14. If

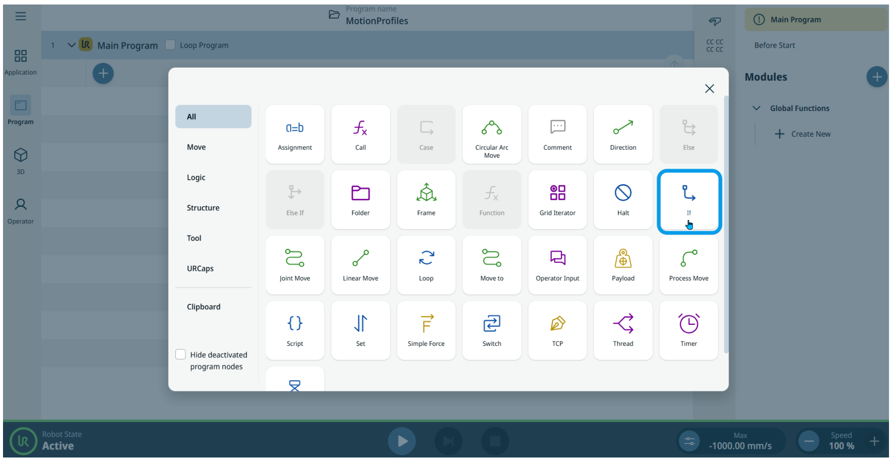
Description

The **If** command changes the robot’s behavior based on sensor inputs or variable values. It allows you to control the flow of your program.

To access If command

1. Go to the main program screen. See Command Nodes.
2. Tap the **If** icon in the Commands toolbox.

   A node is inserted into the program tree, which has an editable field.



3. Tap the **Expression** field.

   The children of the If node will only be executed if the expression evaluates to True.
4. Enter your expression and **Confirm**.

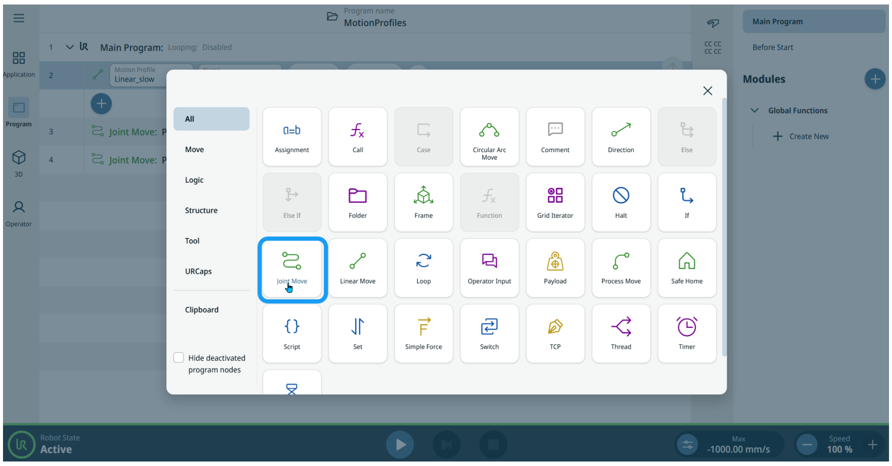
16.3.15. Joint Move

Description

The **Joint Move** is a move command node that supports Motion Profiles. The command creates a movement from point A to point B that is optimal for the robot. The movement may not be a direct line between A and B, but optimal for the start position of the joints and the end position of the joints. Joint Move makes movements that are calculated in the robot arm joint space. Joints are controlled to finish their movements at the same time. This movement type results in a curved path for the tool to follow.

To access Joint Move command

1. Go to the main program screen. See Command Nodes.
2. Tap the **Joint Move** icon in the Commands toolbox.

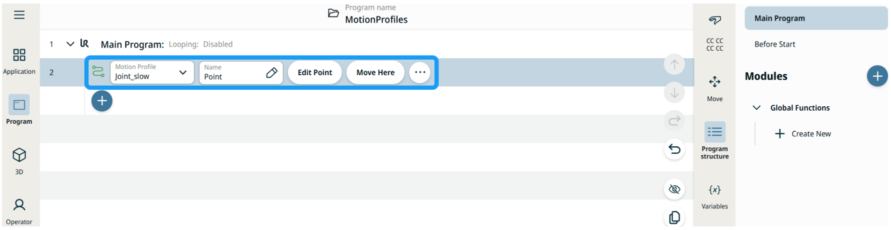


A node with two editable fields, two buttons, and ellipsis icon is inserted into the program tree. These are two editable fields:

- Motion Profile
- Name

These are the two buttons:

- Edit Point
- Move Here



3. Tap the Motion Profile field and select the pertinent from the three choices are available:
   - Custom
   - Joint_fast
   - Joint_slow
4. Tap the Name field, and three tabs appear:
   - Value
   - Variable
   - Expression
5. Enter the data in the **Value**, **Variable**, and **Expression** tabs, and tap **Confirm**. See Value-Variable-Expression Tabs.
6. Tap **Edit Point**. A new screen appears with the robot arm interface where you can jog the robot to the desired position.
7. Tap **More** button.

   More settings screen appears with four settings:
   - Movement
   - Reference
   - Transform
   - Blend

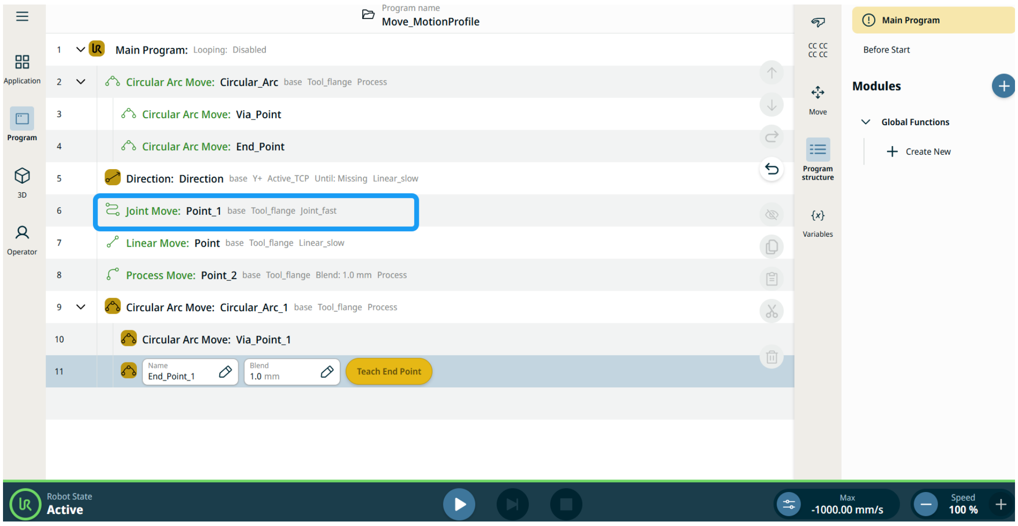
Motion Profile labels on Move Nodes

All **Move-related command nodes** in the program tree display the **Motion Profile** associated with each node. This feature enables users to see the selected Motion Profile on a Move node. This feature is implemented on the following command nodes:

- Linear Move
- Joint Move
- Process Move
- Direction
- 16.3.4 Circular Arc Move on page 126

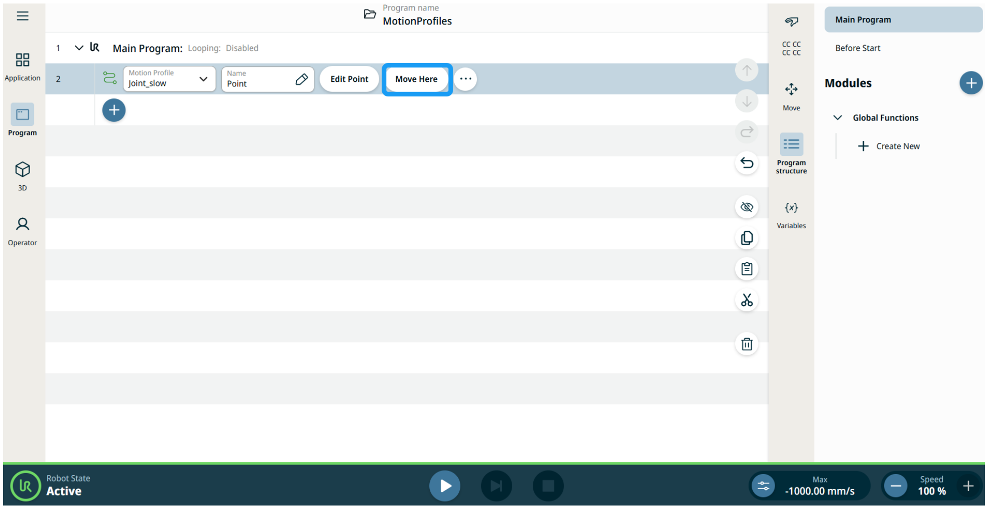
NOTICE

Move nodes with Custom motion profile will display the values for the speed and acceleration instead of Motion profile.

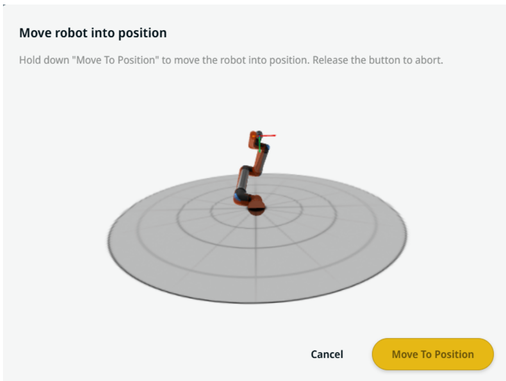

Program name
Move_MotionProfile
Main Program: Looping: Disabled
Circular Arc Move: Circular_Arc base Tool_flange Process
Circular Arc Move: Via_Point
Circular Arc Move: End_Point
Direction: Direction base Y+ Active_TCP Until: Missing Linear_slow
Joint Move: Point_1 base Tool_flange Joint_fast
Linear Move: Point base Tool_flange Linear_slow
Process Move: Point_2 base Tool_flange Blend: 1.0 mm Process
Circular Arc Move: Circular_Arc_1 base Tool_flange Process
Circular Arc Move: Via_Point_1
Name
End_Point_1
Blend
1.0 mm
Teach End Point
Application
Program
3D
Operator
Move
Program structure
Variables
Main Program
Before Start
Modules
Global Functions
Create New
Robot State
Active
Max
-1000.00 mm/s
Speed
100 %

Using the Move Here button

The **Move Here** button is seen in Linear Move and Joint Move commands. This feature enables you to move to the position of a waypoint directly from the program tree. Pressing the **Move Here** button opens the Move robot into position dialog. The button is visible when the waypoint has been taught and is only enabled when the robot status is in **Normal** or **Reduced** mode.

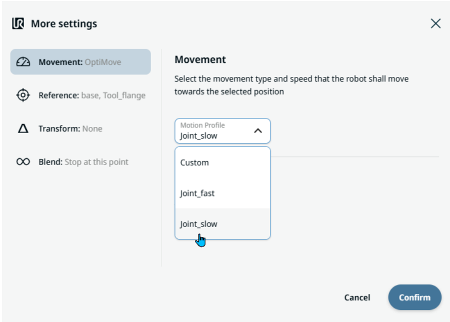


1. Tap the **add icon** to insert the Linear Move and Joint Move in the command tree.
2. Tap the **Edit Point** button of the first move node in the command tree, be it Linear Move or Joint Move nodes.
3. Jog the robot using the plus-minus button in the X, Y, Z coordinates. Tap **Save**.
4. In the second move node in the command tree, tap **Move Here**. A pop-up screen appears to enable **Move robot into position**. Long tap **Move to Position**. Tap **Ok**.



Movement setting

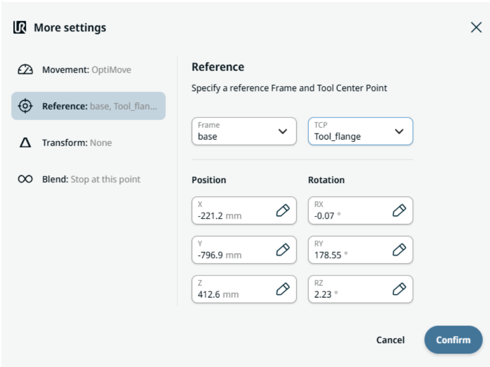
1. Tap the **Movement** setting on the left side. A **Motion Profile** field with the following choices appears on the right side. Joint_slow is the default.
   - Custom
   - Joint_fast
   - Joint_slow



2. If Custom is selected, three more fields appear:
   - Speed Type
   - Speed
   - Acceleration
3. Choose either OptiMove or Classic when you tap Speed Type.
4. Tap the **Speed** field, and the **Value**, **Variable**, and **Expression** tabs pop up.
5. Enter data in each tab, and tap **Confirm**. See Value-Variable-Expression Tabs.
6. Tap the **Acceleration** field, and the **Value**, **Variable**, and **Expression** tabs pop up.
7. Enter data in each tab, and tap **Confirm**. See Value-Variable-Expression Tabs.
8. When **Joint_fast** and **Joint_slow** are selected, the OptiMove speed and acceleration are shown.

Reference setting

1. Tap the **Reference** setting.



2. Choose either **Frame** or **TCP** field.

   In the Frame field, four choices are available:

   - world
   - base
   - flange
   - tcp

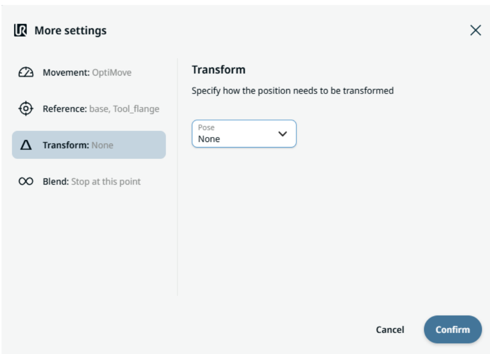
   In the TCP field, two choices are available:

   - Active_TCP
   - Tool_flange

3. In the **Position** column, tap the **X, Y, Z** tabs and enter your chosen value for the robot, and **Confirm**.
4. In the **Rotation** column, tap the **RX, RY, RZ** tabs and enter your chosen value for the robot, and **Confirm**.

Transform setting

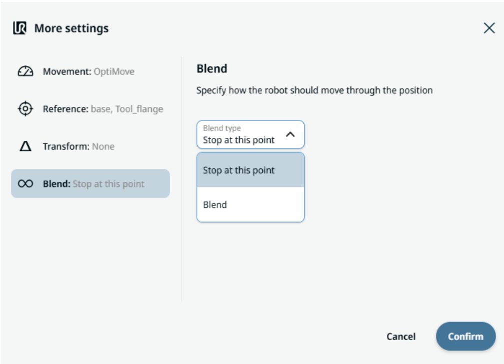
1. Tap the **Transform** setting.



2. Tap the **Pose** field and **Confirm**.

Blend setting

1. Tap the Blend setting.



A **Blend type** field appears with two options:

- Stop at this point
- Blend

2. When you choose **Stop at this point**, no further action is needed but only **Confirm**.
3. Tap Blend, and a **Radius** field is inserted.
4. The **Value**, **Variable**, and **Expression** tabs pop up when you tap the Radius field.
5. Enter data in each tab, and tap **Confirm**. See Value-Variable-Expression Tabs.
6. To finish, tap **Confirm** in the More settings screen.

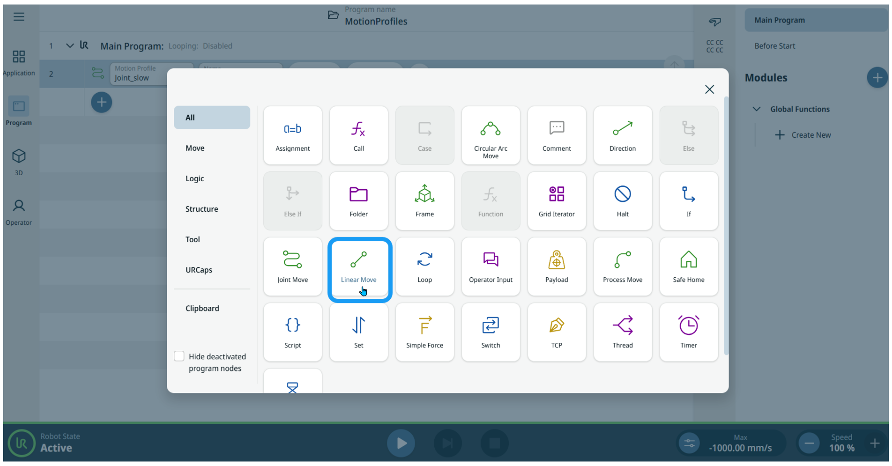
16.3.16. Linear Move

Description

The **Linear Move** command node supports Motions Profiles as Joint Move, but it creates a movement that is a direct line from point A and point B. It moves the Tool Center Point (TCP) linearly between waypoints. This means that each joint performs a more complicated motion to keep the tool on a straight line path.

To access Linear Move command

1. Go to the main program screen. See Command Nodes.
2. Tap the **Linear Move** icon in the Commands toolbox.

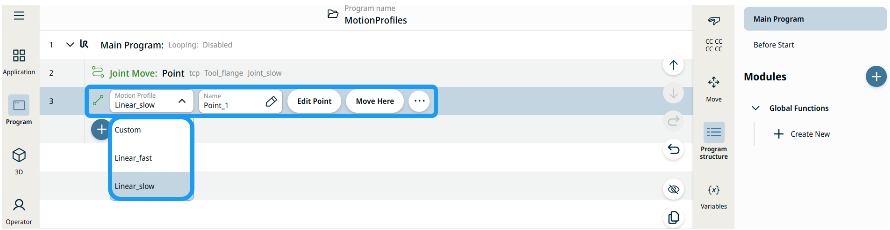


A node with two editable fields, two buttons, and ellipsis icon is inserted into the program tree. These are two editable fields:

- Motion Profile
- Name

These are the three buttons:

- Edit Point
- Move Here

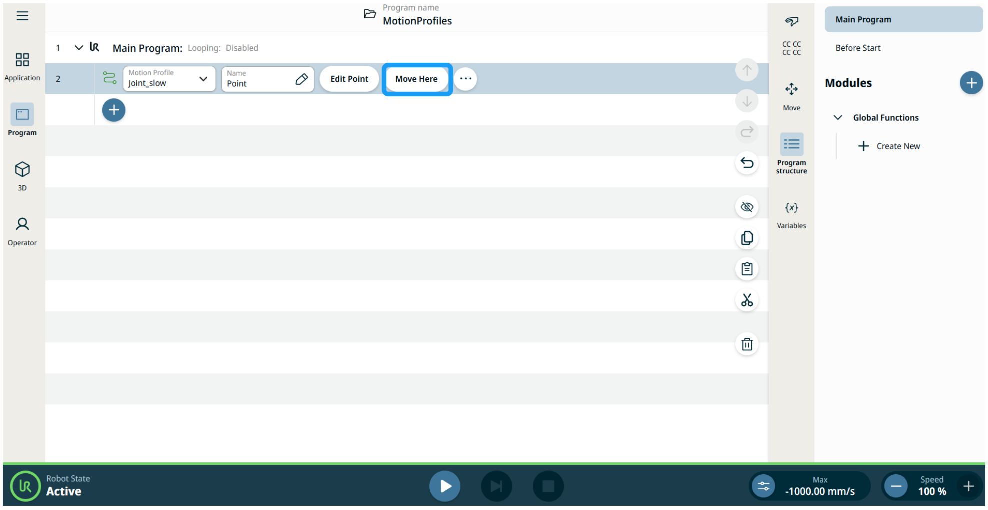


3. Tap the **Motion Profile** field and select the pertinent from the three choices are available:
   - Custom
   - Linear_fast
   - Linear_slow

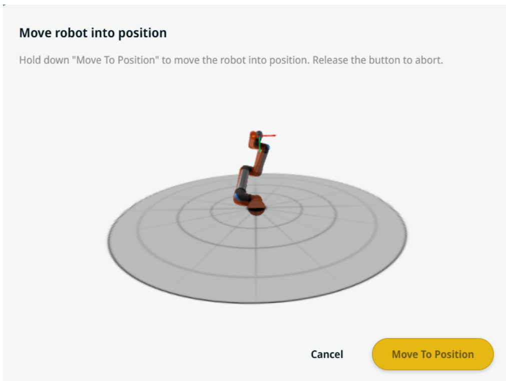
4. Tap the **Name** field, and three tabs appear:
   - Value
   - Variable
   - Expression
5. Enter the data in the **Value**, **Variable**, and **Expression** tabs, and tap **Confirm**. See Value-Variable-Expression Tabs.
6. Tap **Edit Point**. A new screen appears with the robot arm interface.
7. Tap **More** button. The **More settings** screen appears with four settings:
   - Movement
   - Reference
   - Transform
   - Blend

Using the Move Here button

The **Move Here** button is seen in Linear Move and Joint Move commands. This feature enables you to move to the position of a waypoint directly from the program tree. Pressing the **Move Here** button opens the Move robot into position dialog. The button is visible when the waypoint has been taught and is only enabled when the robot status is in **Normal** or **Reduced** mode.



1. Tap the **add icon** to insert the Linear Move and Joint Move in the command tree.
2. Tap the **Edit Point** button of the first move node in the command tree, be it Linear Move or Joint Move nodes.
3. Jog the robot using the plus-minus button in the X, Y, Z coordinates. Tap **Save**.
4. In the second move node in the command tree, tap **Move Here**. A pop-up screen appears to enable **Move robot into position**. Long tap **Move to Position**. Tap **Ok**.



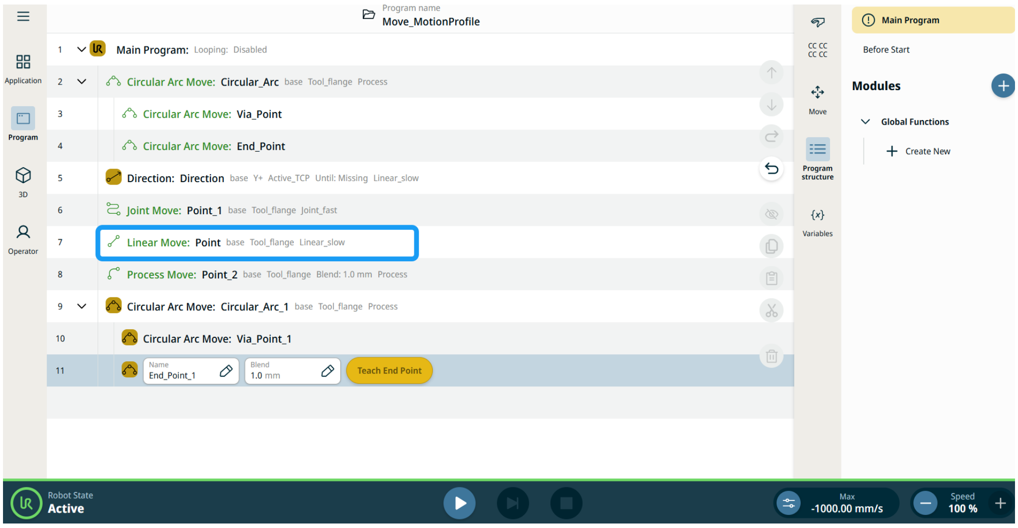
Motion Profile labels on Move Nodes

All **Move-related command nodes** in the program tree display the **Motion Profile** associated with each node. This feature enables users to see the selected Motion Profile on a Move node. This feature is implemented on the following command nodes:

- Linear Move
- Joint Move
- Process Move
- Direction
- 16.3.4 Circular Arc Move on page 126

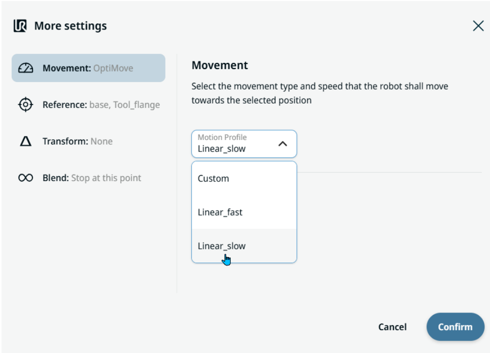
NOTICE

Move nodes with Custom motion profile will display the values for the speed and acceleration instead of Motion profile.



Movement setting

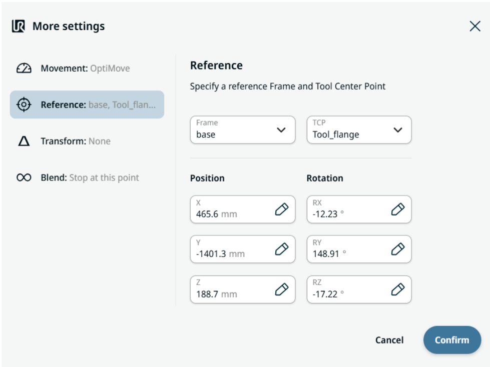
1. Tap the **Movement** setting on the left side. A **Motion Profile** field with the following choices appears on the right side. **Linear_slow** is the default.
   - Custom
   - Linear_fast
   - Linear_slow



2. If **Custom** is selected, three more fields appear:
   - Speed Type
   - Speed
   - Acceleration
3. Choose either OptiMove or Classic when you tap Speed Type.
4. Tap the **Speed** field, and the **Value**, **Variable**, and **Expression** tabs pop up.
5. Enter data in each tab, and tap **Confirm**. See Value-Variable-Expression Tabs.
6. Tap the **Acceleration** field, and the **Value**, **Variable**, and **Expression** tabs pop up.
7. Enter data in each tab, and tap **Confirm**. See Value-Variable-Expression Tabs.
8. When **Linear_fast** and **Linear_slow** are selected, the OptiMove speed and acceleration are shown.

Reference setting

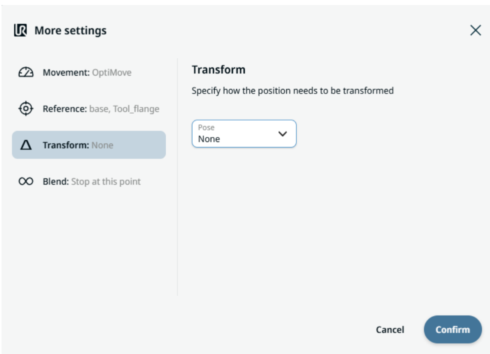
1. Tap the **Reference** setting.



2. Choose either **Frame** or **TCP** field.
3. In the Frame field, four choices are available:
   - world
   - base
   - flange
   - tcp
4. In the TCP field, two choices are available:
   - Active_TCP
   - Tool_flange
5. In the **Position** column, tap the **X, Y, Z** tabs and enter your chosen value for the robot, and **Confirm**.
6. In the **Rotation** column, tap the **RX, RY, RZ** tabs and enter your chosen value for the robot, and **Confirm**.

Transform setting

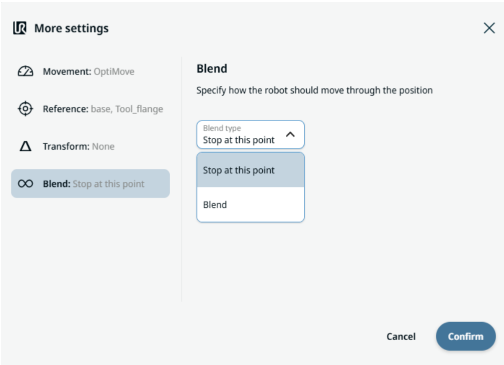
1. Tap the **Transform** setting.



2. Tap the **Pose** field and **Confirm**.

Blend setting

1. Tap the Blend setting.



   A **Blend type** field appears with two options:
   - Stop at this point
   - Blend
2. When you choose **Stop at this point**, no further action is needed but only **Confirm**.
3. Tap Blend, and a **Radius** field is inserted.
4. The **Value**, **Variable**, and **Expression** tabs pop up when you tap the Radius field.
5. Enter data in each tab, and tap **Confirm**. See Value-Variable-Expression Tabs.
6. To finish, tap **Confirm** in the More settings screen.

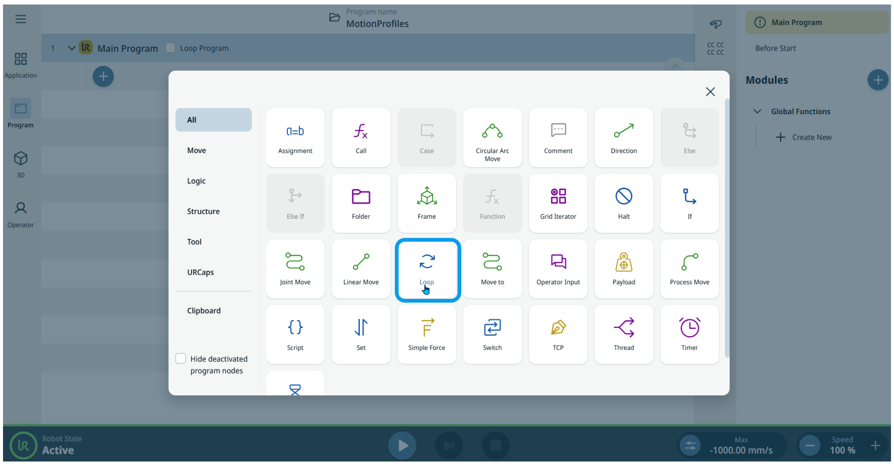
16.3.17. Loop

Description

The **Loop** command enables you to loop underlying program commands in PolyScope X.

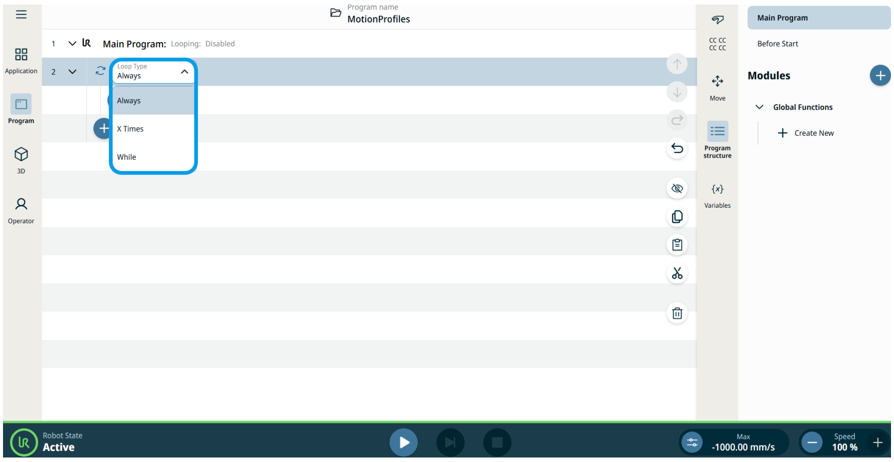
To access Loop command

1. Go to the main program screen. See Command Nodes.
2. Tap the **Loop** icon in the Commands toolbox.



A node with **Loop Type** field is inserted into the program tree.

3. Tap the **Loop Type** field. Three choices are available:
   - **Always**. Any commands under the loop command will be repeated continuously, such as the **Loop Program**, which continually repeats everything under the **Main Program**.
   - **X Times**. Repeats any commands in the loop command the number of times that's entered.
   - **While**. Use some type of expression as the trigger to end the loop cycle.



4. Tap the **Always** tab. No further action to do.
5. Tap **X Times**, and two additional tabs appear on the right:
   - X Times
   - Variable Name
6. On the **X Times** tab, edit and **Confirm**.
7. On the **Variable Name** tab, edit and **Confirm**.
8. Tap **While**. An additional **Expression** tab appears on the right.
9. Tap **Expression** to create expression that must be between 1 and 1000 characters, and **Confirm**.

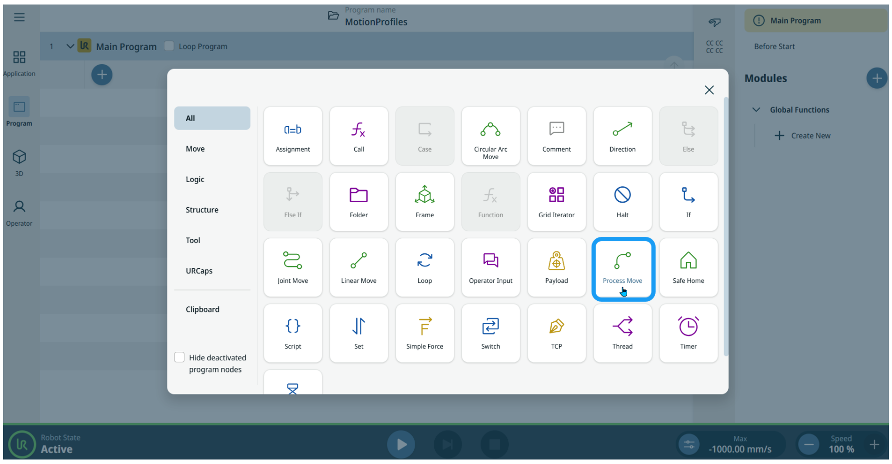
16.3.18. Process Move

Description

The **Process Move** command enables you to program a linear movement with constant speed through a number of points. You must set blend radius to make sure that the robot moves at a constant speed between each move node. You can specify a reference frame and a transform pose.

To access Process Move command

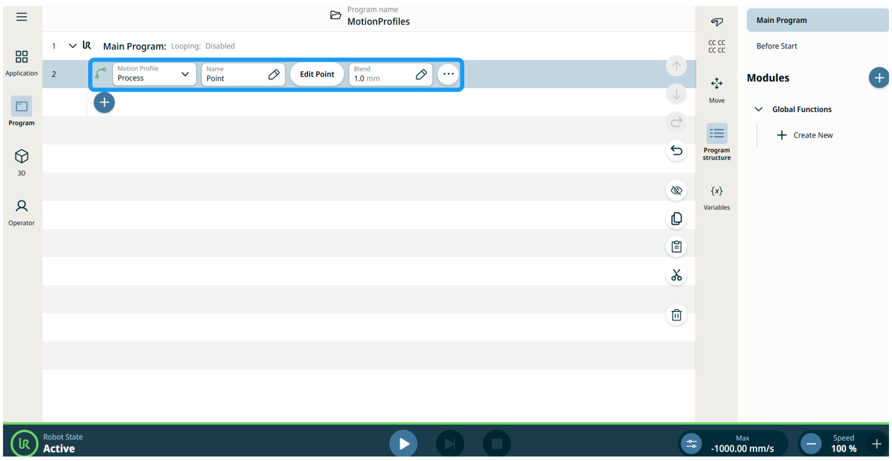
1. Go to the main program screen. See Command Nodes.
2. Tap the **Process Move** icon in the Commands toolbox.



A node is inserted into the program tree. The node has three editable fields, Edit Point button, and ellipsis icon.

These are the three fields:

- Motion Profile
- Name
- Blend



3. In the **Motion Profile** field, you can choose either **Custom** or **Process**.

   Custom motion profile enables you to define movement data in **Joint Move and Linear Move** nodes.

   Process motion profile is a predefined motion in **Joint Move and Linear Move** nodes.
4. Tap the **Name** field.

   Three tabs appear:
   - Value
   - Variable
   - Expression
5. Enter data in the **Value**, **Variable**, and **Expression** tabs, and tap **Confirm**. See Value-Variable-Expression Tabs.
6. Tap **Edit Point**.

   A new screen appears with the robot arm interface.
7. Tap the **Blend** field.

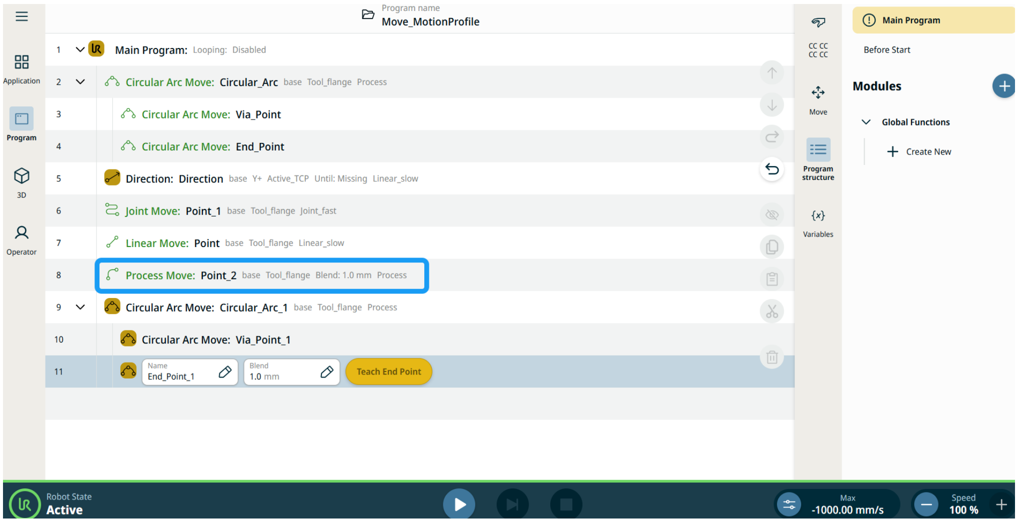
   Three tabs appear:
   - Value
   - Variable
   - Expression
8. Enter data in the **Value**, **Variable**, and **Expression** tabs, and tap **Confirm**. See Value-Variable-Expression Tabs.

Motion Profile labels on Move Nodes

All **Move-related command nodes** in the program tree display the **Motion Profile** associated with each node. This feature enables users to see the selected Motion Profile on a Move node. This feature is implemented on the following command nodes:

- Linear Move
- Joint Move
- Process Move
- Direction
- 16.3.4 Circular Arc Move on page 126

> **NOTICE**
> Move nodes with Custom motion profile will display the values for the speed and acceleration instead of Motion profile.

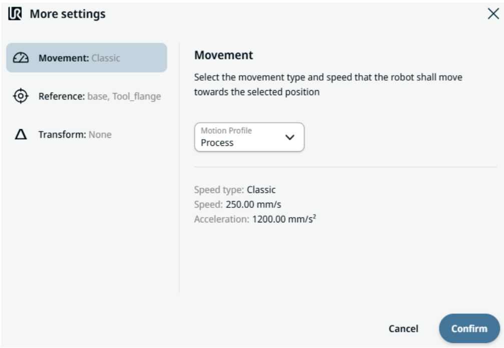

Program name
Move_MotionProfile
Main Program: Looping: Disabled
Circular Arc Move: Circular_Arc base Tool_flange Process
Circular Arc Move: Via_Point
Circular Arc Move: End_Point
Direction: Direction base Y+ Active_TCP Until: Missing Linear_slow
Joint Move: Point_1 base Tool_flange Joint_fast
Linear Move: Point base Tool_flange Linear_slow
Process Move: Point_2 base Tool_flange Blend: 1.0 mm Process
Circular Arc Move: Circular_Arc_1 base Tool_flange Process
Circular Arc Move: Via_Point_1
Name
End_Point_1
Blend
1.0 mm
Teach End Point
Application
Program
3D
Operator
Move
Program structure
Variables
Main Program
Before Start
Modules
Global Functions
Create New
Robot State
Active
Max
-1000.00 mm/s
Speed
100 %

To use More Option

1. Tap the **ellipsis** icon.

   More settings screen appears with three settings:

   - Movement
   - Reference
   - Transform



2. Tap the **Movement** setting on the left side.

   A **Motion Profile** field with selections **Custom** and **Process** is on the right side.
3. If Custom is selected, a **Speed** field and **Acceleration** field appears.
4. Tap the **Speed** field, and the **Value**, **Variable**, and **Expression** tabs pop up.
5. Enter data in each tab, and tap **Confirm**. See Value-Variable-Expression Tabs.
6. Tap the **Acceleration** field, and the **Value**, **Variable**, and **Expression** tabs pop up.
7. Enter data in each tab, and tap **Confirm**. See Value-Variable-Expression Tabs.
8. Choose **Process** motion profile and tap **Confirm**.
9. Tap the **Reference** setting.
10. Choose either **Frame** or **Set TCP** field.
11. In the **Position** column, tap the **X, Y, Z** tab and enter your chosen value for the robot, and **Confirm**.
12. In the **Rotation** column, tap the **RX, RY, RZ** tab and enter your chosen value for the robot, and **Confirm**.
13. In the **Transform** setting, tap the **Pose** field and **Confirm**.

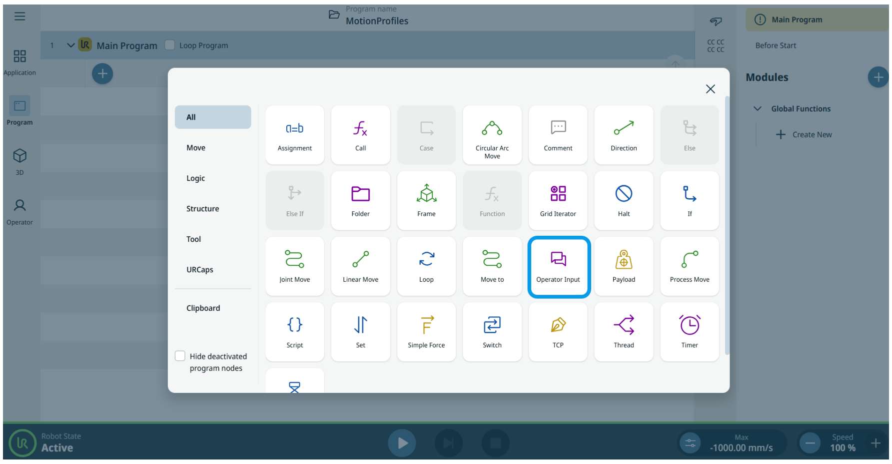
16.3.19. Operator Input

Description

The **Operator Input** command is used to request input from the operator or display a message during program execution. The command allows you to define a question, select the input type, and store the response in a variable.

To access Operator Input command

1. Go to the main program screen. See Command Nodes.
2. Tap the **Operator Input** icon in the Commands toolbox.



A node with four editable fields is inserted into the program tree.

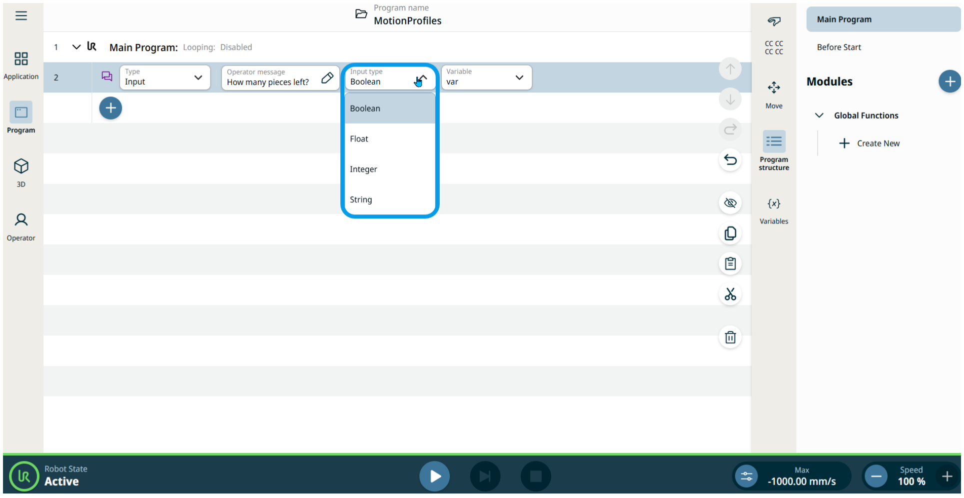
The editable fields are:

- Type
- Operator message
- Input type
- Variable Name

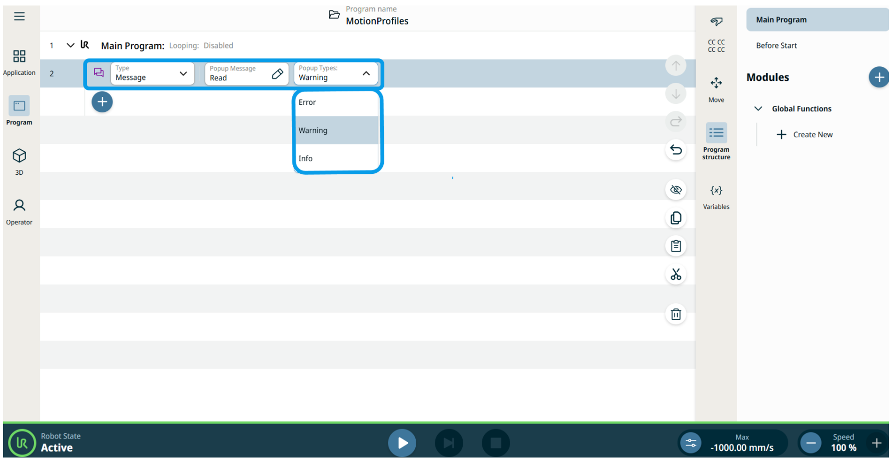
3. Tap the **Type** field and select either **Input** or **Message**.
4. When you select **Input**, enter and edit your note or message in the **Operator message** field and tap **Confirm**.

   The program stops at this point, displays the message from the Message field, and waits for a response.

   The type of response is determined by the Input Type field.
5. Tap the **Input type** field.



6. Select one of the four input types given:
   - **Boolean** - Select Yes/No buttons.
   - **Float** - Enter a decimal value.
   - **Integer** - Enter a whole number.
   - **String** - Enter text.
7. Tap the **Variable Name** field.
8. Tap the **Edit** tab inside the field to edit the variable name of your preference.
9. When you select **Message** in the **Type** field, the node in the program tree modifies from having four editable fields to two:
   - Popup Message
   - Popup Types



10. Tap **Popup Message** field.
11. Create your message and tap **Confirm**.
12. Tap **Popup Types** field.
13. Select the type from the three choices:
    - Error
    - Warning
    - Info

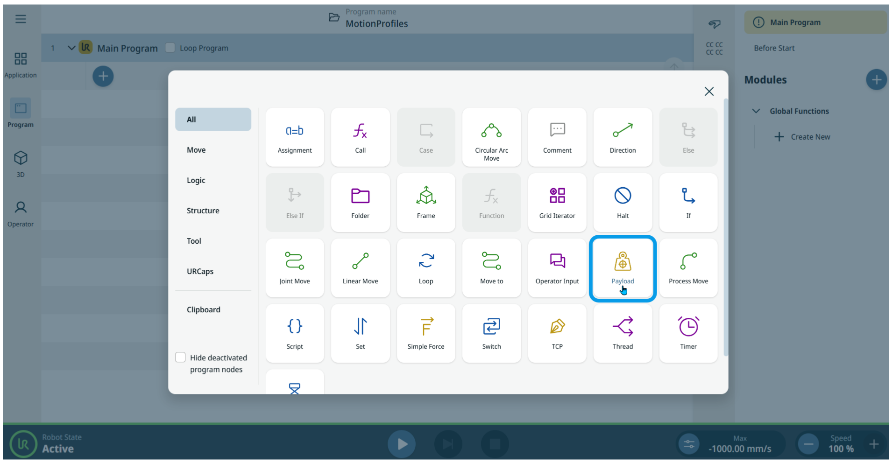
16.3.20. Payload

Description

The **Payload** command allows you to configure the payload of the robot, which is the combined weight of everything attached to the robot tool flange, and the center of gravity to predefined values or custom ones.

To access Payload command

1. Go to the main program screen. See Command Nodes.
2. Tap the **Payload** icon in the Commands toolbox.



A node with an editable **Type** field is inserted into the program tree. The fields are:

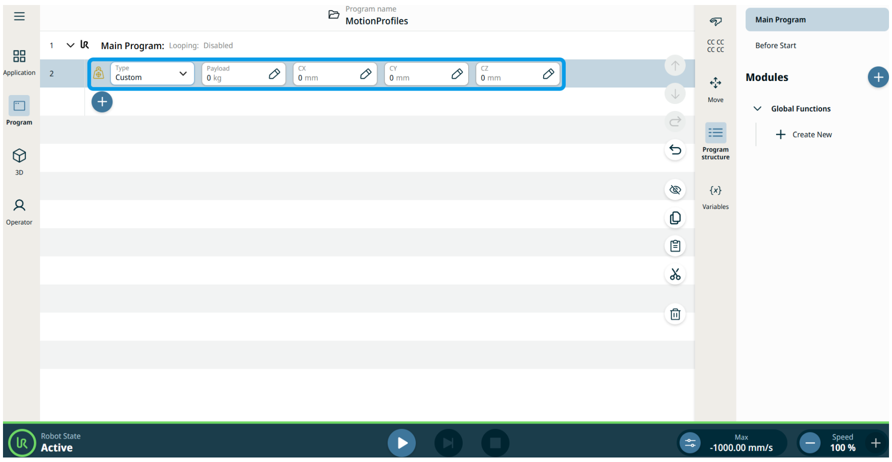
- Custom
- Robot

3. Tap **Custom**.

When this is selected, the payload and center of gravity can be directly set.

Four additional fields appear on the right side:

- Payload
- CX
- CY
- CZ



4. Tap the **Payload** field.

   This contains a list of all end effectors defined in the End Effectors application inside the Application menu, which includes the tool flange predefined end effector. See End Effectors Application.

   Three tabs to input the payload are available:

   - Value
   - Variable
   - Expression

   Note: The Variable tab is inactive.

5. Make the input in each tab and tap **Confirm**. See Value-Variable-Expression Tabs.
6. Tap the **CX** field and repeat step 5.
7. Tap the **CY** field and repeat step 5.
8. Tap the **CZ** field and repeat step 5.

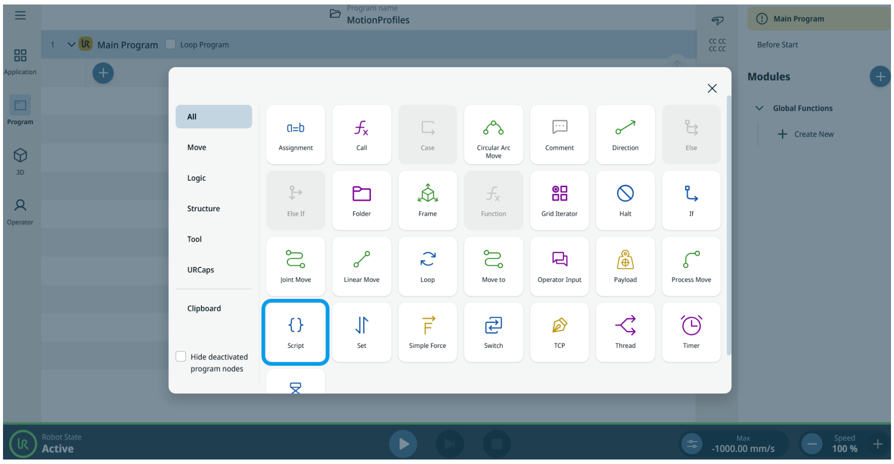
16.3.21. Script

Description

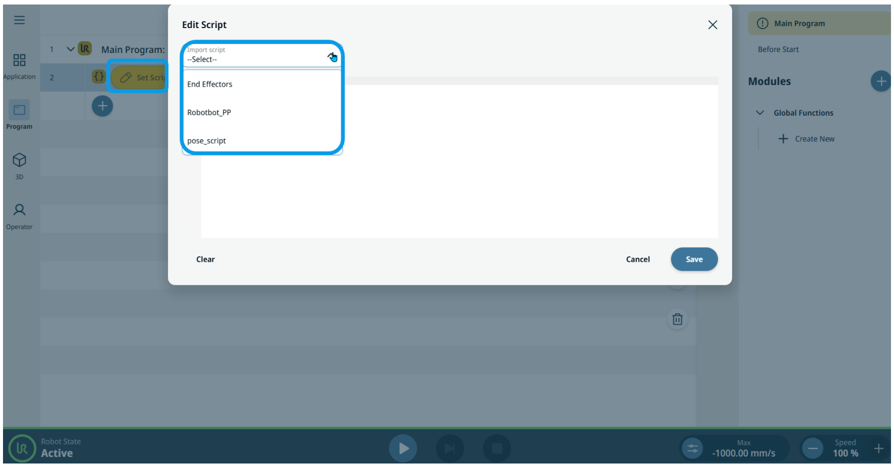
The **Script** command enables you to write and insert a URscript code or files directly in your program.

To access Script command

1. Go to the main program screen. See Command Nodes.
2. Tap the **Script** icon in the Commands toolbox. A node is inserted into the program tree with a **Set Script** button.



3. Tap the **Set Script** button. A new screen appears where you can edit the script. Enter the script and tap **Save**.



4. Tap the **Import script** field to import a script from the Script Manager, as well as to clear the contents. See Script Files.

> **NOTICE**
>
> The script file is copied from the script manager into the script node when inserted. Any future imports of the file into the script manager will not change the existing programs. The programs will need to be manually updated, if necessary.

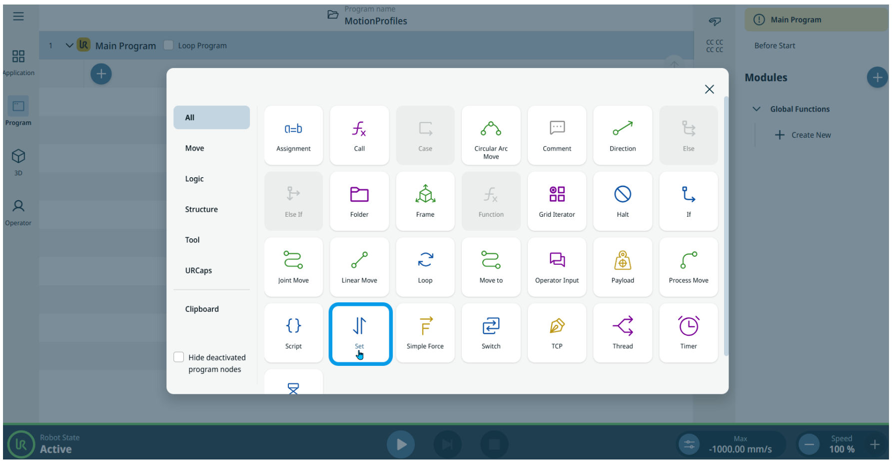
16.3.22. Set

Description

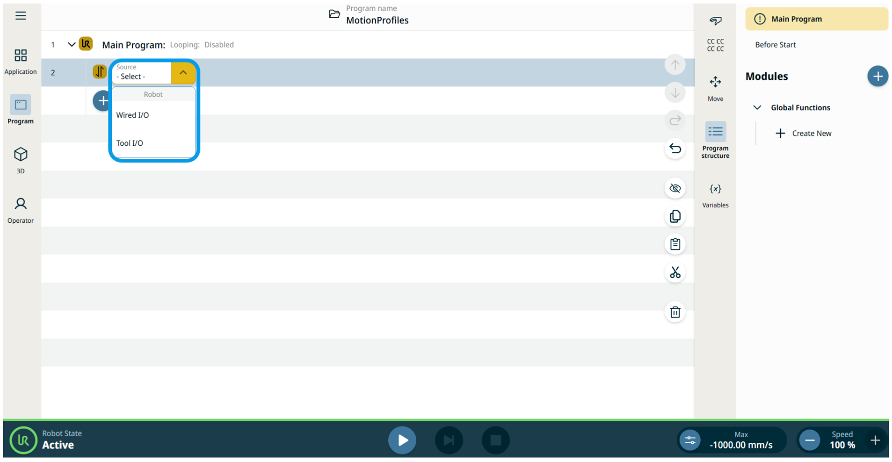
The **Set** command enables you to set a given value to either digital or analog outputs.

To access Set command

1. Go to the main program screen. See Command Nodes.
2. Tap the **Set** icon in the Commands toolbox.



3. A node with an editable **Source** field is inserted into the program tree that enables you to select **Wired IO** and **Tool IO**.
4. Tap the **Source** field.



5. When you select **Wired IO**, a **Signal** field is inserted.
6. In the **Signal** field, tap your preferred signal.
   - DO 0-DO 7: Digital signal
   - CO 0-CO 7: Digital signal
   - AO 0-AO 1: Analog signal
7. When a digital signal is selected, a **Value** field is inserted to the right.
8. Choose either **High** or **Low**.
9. When an analog signal is selected, a **Value** field is inserted to the right.
10. Enter data in the **Value**, **Variable**, and **Expression** tabs, and tap **Confirm**. See Value-Variable-Expression Tabs.
11. When you select **Tool IO**, a **Signal** field is inserted.
12. Choose either DO 0 or DO 1.

    A **Value** field is inserted to the right with three options:
    - Use Variable
    - High
    - Low
13. Tap **Use Variable**, and the **Variable** and **Expression** tabs appear.
14. Enter data in the two tabs, and tap **Confirm**. See Value-Variable-Expression Tabs.
15. You can also choose either **High** or **Low** in the Value field.

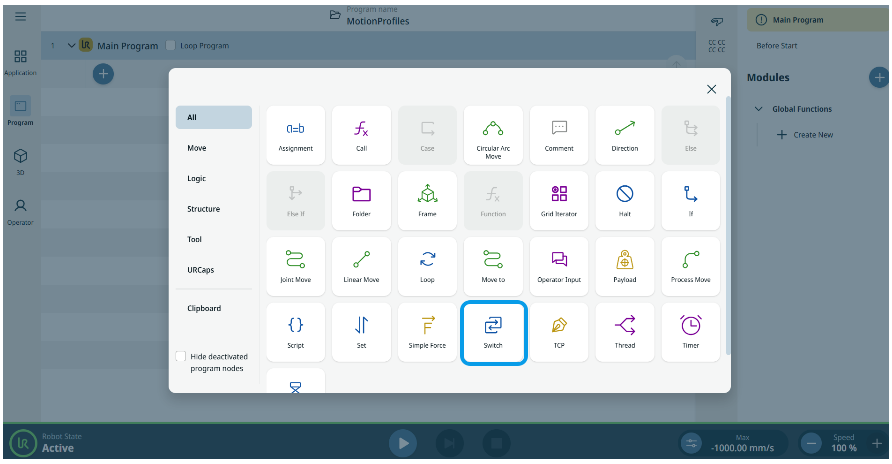
16.3.23. Switch

Description

The **Switch** command enables you to make the robot change behavior based on sensor inputs or variable values.

To access Switch command

1. Go to the main program screen. See Command Nodes.
2. Tap the **Switch** icon in the Commands toolbox.

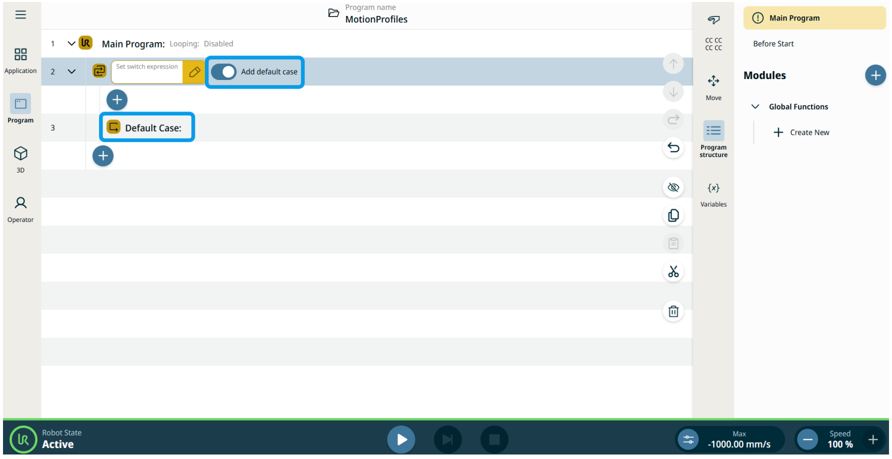


A node with **Set switch expression** field and **Add default case** button is inserted into the program tree.

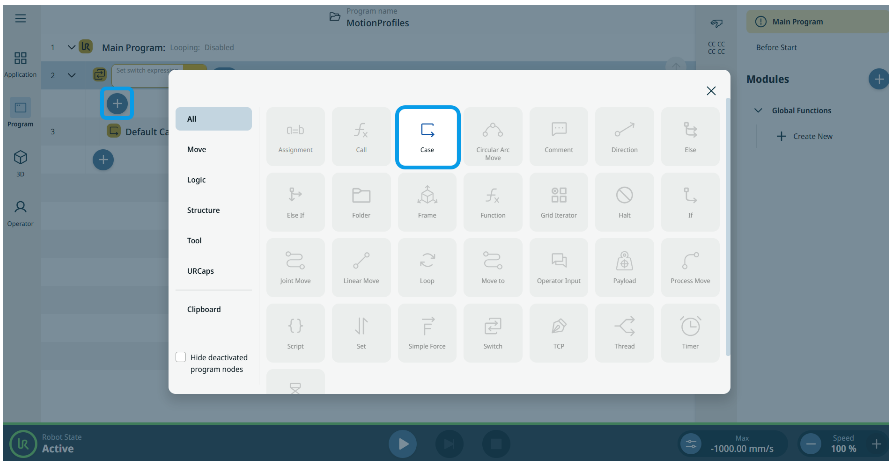
3. Tap the **Set switch expression** field, and two options are available:
   - Variable
   - Expression
4. Tap **Variable** and select or switch based on an existing variable or an expression.

   Note: Open the Assignment command node to activate the Variable tab in Switch node.
5. Enter the expression in **Expression** tab and **Confirm**.

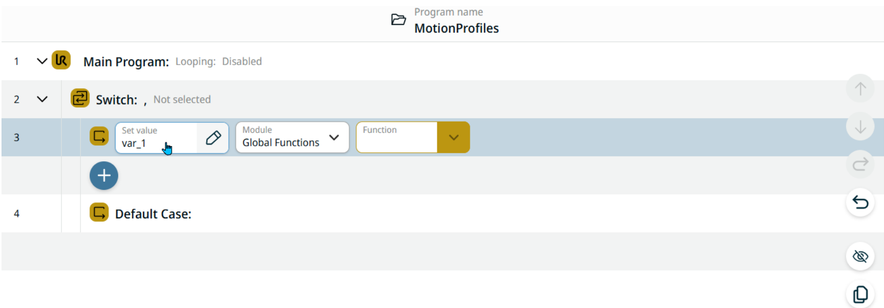
6. Tap **Add default case** button, and **Default Case** is inserted.



7. Tap the **add icon** inside the Switch node to add a case node.
8. Tap the **Case** icon in the Commands toolbox. See Case.



Each case checks the switch variable or expression against a single value. If a case and switch match, it calls the function you specify in the case node.



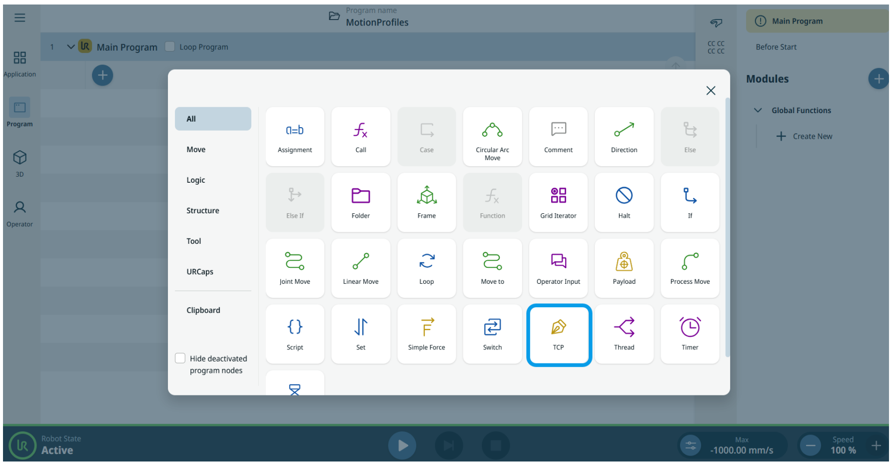
16.3.24. TCP

Description

The **TCP** command enables you to change the active Tool Center Point (TCP) during program execution. It uses the TCP already defined in the End Effectors application inside the Application menu. See End Effectors Application.
The TCP command updates the active TCP when changes happen mid-program to ensure the robot's motion planning stays accurate.

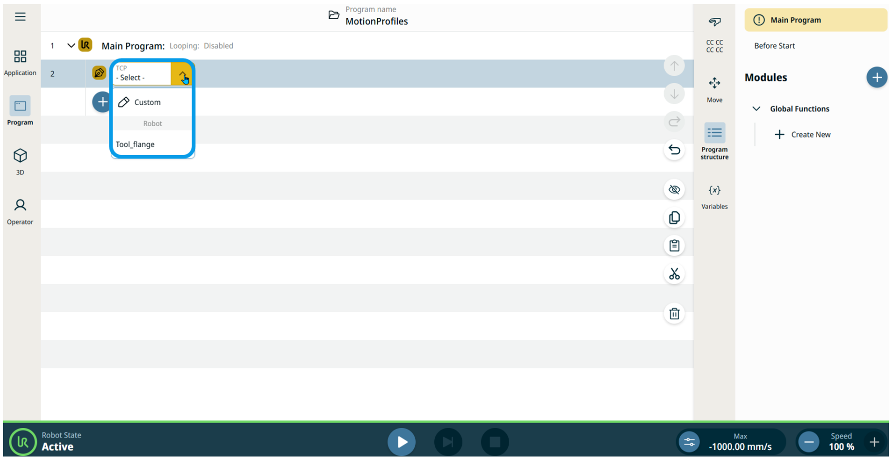
To access TCP command

1. Go to the main program screen. See Command Nodes.
2. Tap the **TCP** icon in the Commands toolbox.



A node with an editable **TCP** field is inserted into the program tree.

3. Tap the **TCP** field and select your preference.

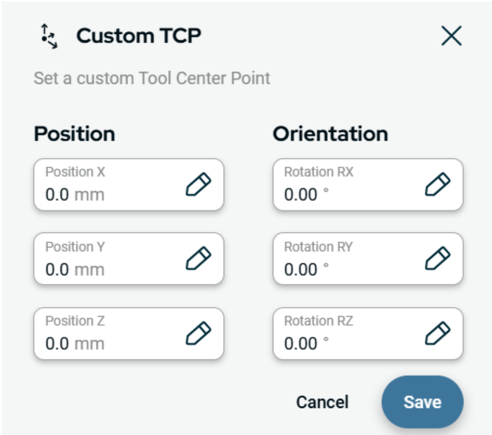


4. Select **Custom**.

   A **Setup** tab is inserted to the right.

5. Tap the **Setup** tab.

   The **Custom TCP** screen appears.



6. Tap the **Position X, Position Y, Position Z** tabs in the **Position** column, and enter the Value, Variable, and Expression on their respective tabs, and **Confirm**. See Value-Variable-Expression Tabs.

7. Tap the **Rotation RX, Rotation RY, Rotation RZ** tabs in the **Orientation** column, and enter the Value, Variable, and Expression on their respective tabs, and **Confirm**. See Value-Variable-Expression Tabs.

8. Tap **Save**.

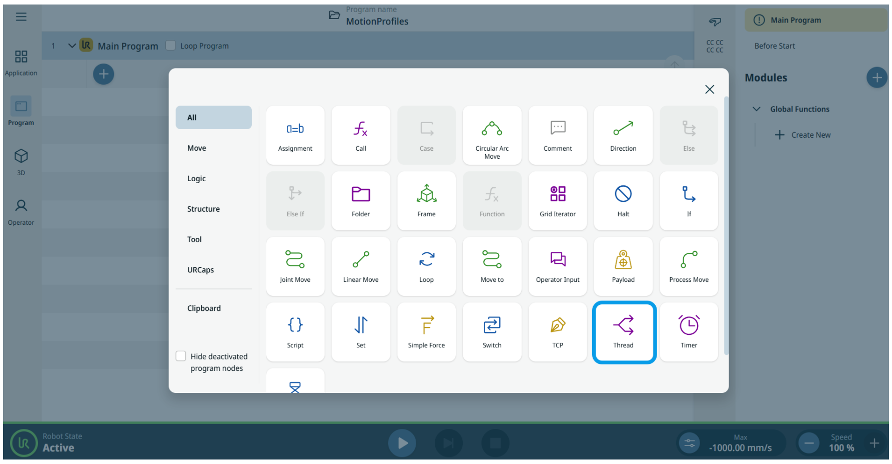
16.3.25. Thread

Description

The **Thread** command enables you to control an external machine independently of the robot arm. A thread can communicate with the robot program with variables and output signals.

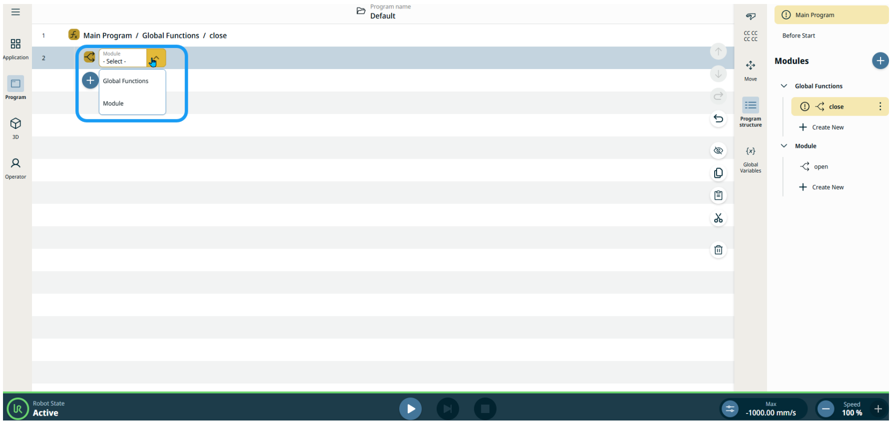
To access Thread command

1. Go to the main program screen. See Command Nodes.
2. Tap the **Thread** icon in the Commands toolbox.



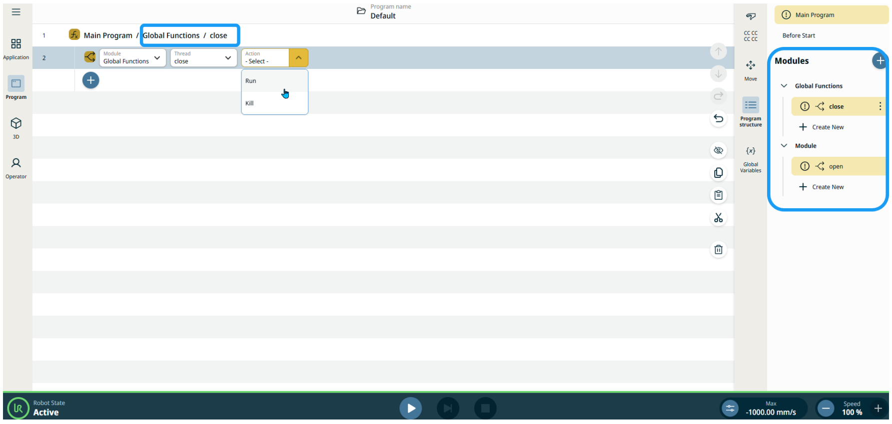
A node with a **Module** field is inserted into the program tree.

3. Tap the **Module** field, and two selections are given:
   - Global Functions
   - Module

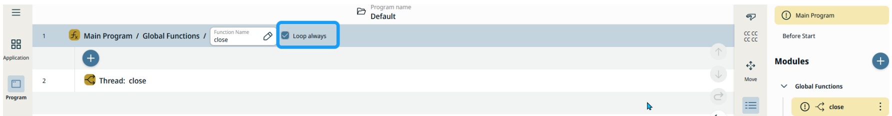


4. Select either **Global Functions** or **Module**, and a **Thread** field appears at the right.

5. Note: For the Thread field to be active and selectable, you first have to create new global functions and modules in the Program Structure and marked as thread. See Modules and Global Functions.



The function/module is now marked as a thread, and tap the **Loop always** checkbox to make the thread execution loop when done.



6. Tap the **Thread** field and select the function or module you created. An **Action** field is inserted further.
7. Tap the **Action** field and select **Run** or **Kill** to start or stop the thread execution at that point.

   Threads that are meant to run for the entire duration of the program execution can be started from **Before Start** section of the **Main Program**.

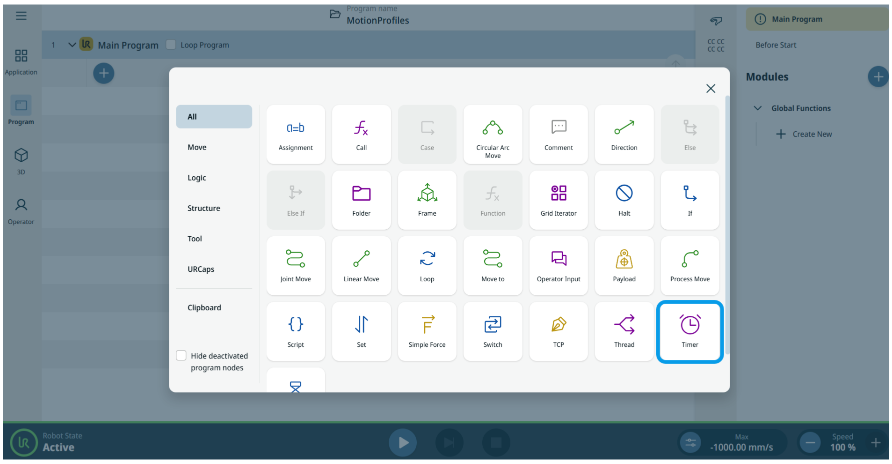
16.3.26. Timer

Description

The **Timer** command is used to measure real time during program execution. Each timer node adds a timer variable or references an existing timer variable.

To access Timer command

1. Go to the main program screen. See Command Nodes.
2. Tap the **Timer** icon in the Commands toolbox.



A node with two editable fields, **Variable** and **Action**, is inserted into the program tree.

3. Tap the **Variable** field.
4. Tap **Rename** if you choose a new name, and **Confirm**.
5. Tap the **Action** field.

   Three options are available:

   - Start
   - Pause
   - Reset

   Note: Timer can be started, paused, and reset to 0.

   The elapsed time since the timer started can be read from the timer variable.

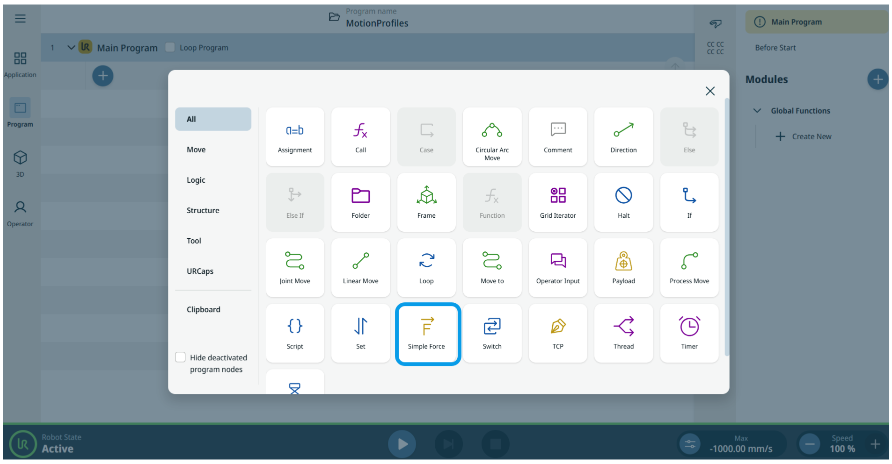
16.3.27. Simple Force

Description

The **Simple Force** command enables you to use only one required axis for a desired force. The force along this axis is adjustable and is always applied along the z-axis of the selected feature.

To access Simple Force command

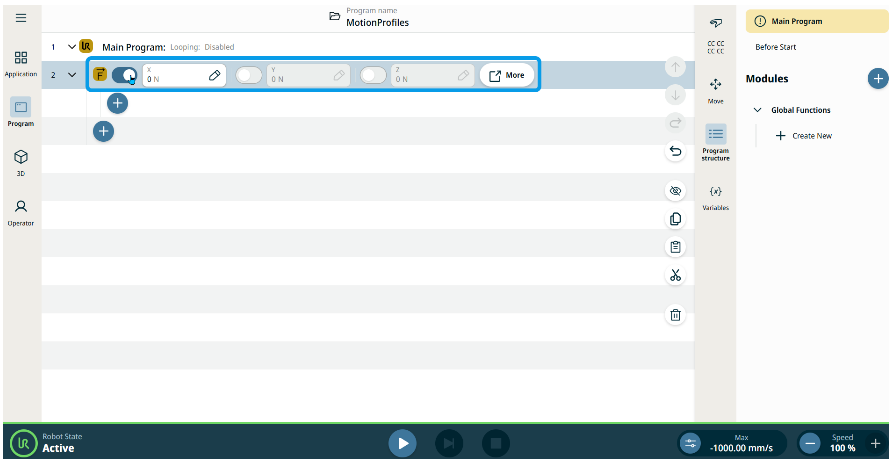
1. Go to the main program screen. See Command Nodes.
2. Tap the **Simple Force** icon in the Commands toolbox.



A node is inserted into the program tree. The node has three editable fields and a **More** button for advanced options.

The three fields are **X**, **Y**, and **Z**. These are inactive by default.

Each field has a slider button on the left side to switch activate or deactivate.



3. Tap the slider button to the right to activate the three fields.

   Three tabs to input the coordinates are available:

   - Value
   - Variable
   - Expression

   Note: The Variable tab is inactive.
4. On the **Value** tab, enter your preferred value and **Confirm**.
5. Tap the **Expression** tab and enter the value and **Confirm**.

   See Value-Variable-Expression Tabs.

To use More options

1. Tap the **More** button.

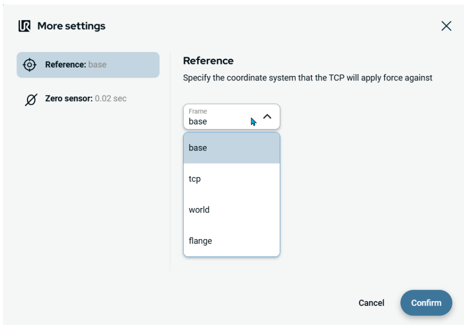
   A new screen appears containing two advanced settings:

   - Reference
   - Zero sensor

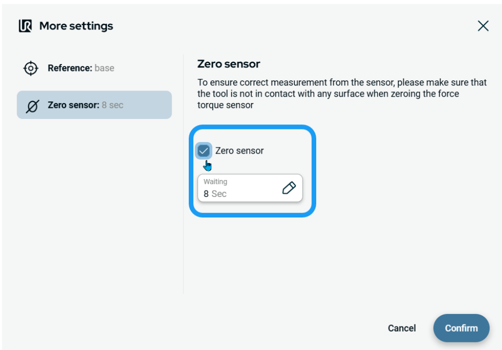
2. Tap **Reference** on the left side.

   A new **Frame** field appears in the right side, containing four selections:

   - base
   - tcp
   - world
   - flange



3. Select your chosen frame of your reference and **Confirm**.
4. Tap **Zero sensor** on the left side.



5. Tap the Zero sensor **check box** to activate or deactivate the setting.
6. When Zero sensor is activated, tap the **Waiting** field below.
7. Enter the waiting **Value** in seconds.
8. Tap **Confirm**.

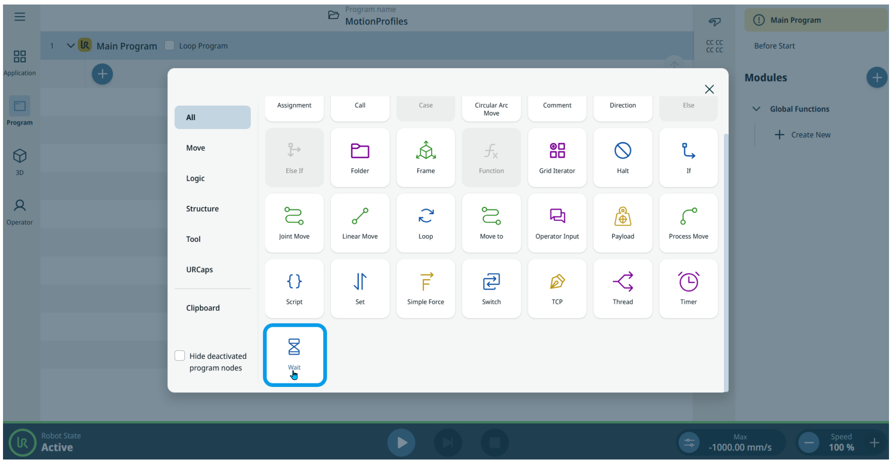
16.3.28. Wait

Description

The **Wait** command pauses the robot's movement when new inputs are introduced into the program. You can add a Wait command to a program with external sensors to make the robot wait for one of the sensors to activate before the program continues.

To access Wait command

1. Go to the main program screen. See Command Nodes.
2. Tap the **Wait** icon in the Commands toolbox.



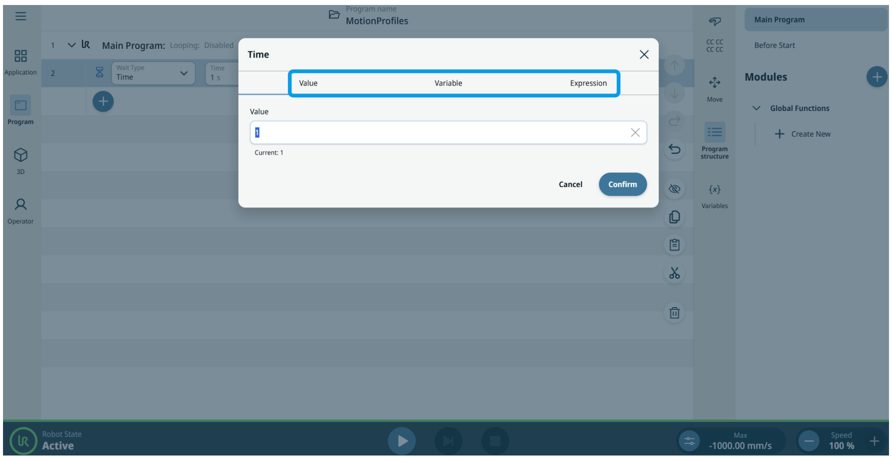
A node with two editable fields is inserted into the program tree:

- Wait Type
- Time

3. Tap the **Wait Type** field and choose **Time**.

   When you select this, the program waits for a number of specified seconds.

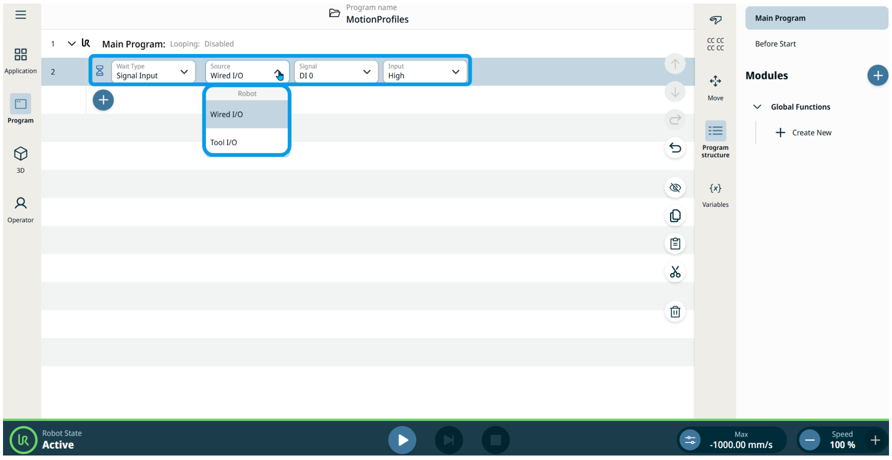
4. On the **Time** field, enter the **Value** and **Expression**, and tap **Confirm**. See Value-Variable-Expression Tabs.



5. Tap the **Wait Type** field and choose **Signal Input**.

   When you choose this, additional fields are added to the right for you to specify the type of input.

6. Tap the **Source** field, and two standard inputs are available:
   - Wired IO
   - Tool IO



7. When you choose either **Wired IO** or **Tool IO**, a **Signal** field is inserted to the right.
8. In the **Signal** field, tap your preferred signal.
   - Digital signal: DI 0-DI 7
   - Digital signal: CI 0-CI 7
   - Analog signal: AI 0-AI 1

   When a digital signal is selected, an **Input** field is inserted to the right.
9. Choose either **High** or **Low**.

   When an analog signal is selected, **Operator** and **Input** fields are inserted to the right.
10. Tap the **Operator** field.
11. Choose either **<** or **>** to specify the wait compared to a value.
12. Edit your **Input** field and **Confirm**.

17. First Program

Description

The first program creation in PolyScope X enables you to teach the robot arm how to move using a series of waypoints to set up a path for it to follow.

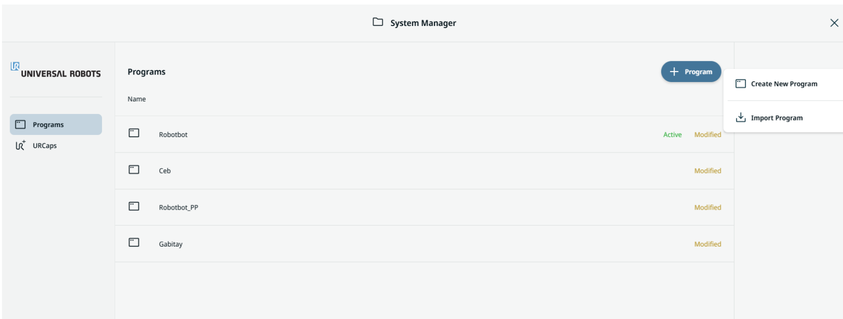
To create a simple program

For this simple program, you (1) move to position A using a joint move type, (2) set an IO, (3) wait for 3 seconds, (4) unset the IO, (5) move to position B using again a joint move type, and (6) repeat the movement.

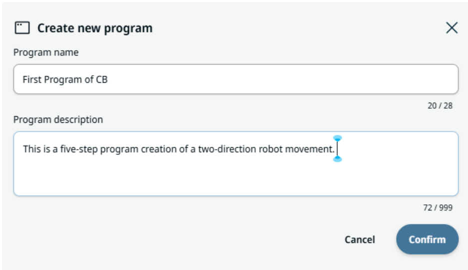
> **NOTICE**
>
> 1. Do not drive the robot into itself or anything else as this may cause damage to the robot.
> 2. This is only a quick start guide to show how easy it is to use a UR robot. It assumes a harmless environment and a very careful user. Do not increase the speed or acceleration above the default values. Always conduct a risk assessment before placing the robot into operation.

System Manager access

1. In the main screen, tap the **header** to open the System Manager.
2. Tap the **add Program** icon in the **System Manager** screen and choose **Create New Program**.



3. Enter the **program name** and **description** in the two fields provided in the **Create new program** box. Tap **Confirm**.

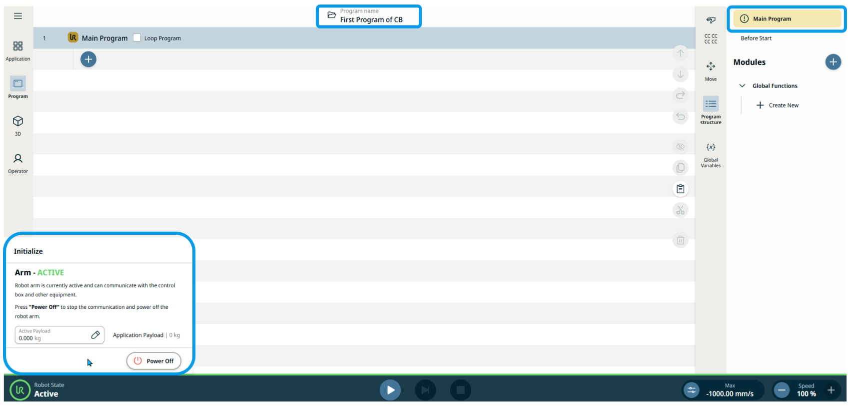


The main screen now shows your program name. At the multitask screen, you see the **Main Program** tab in yellow, indicating that program creation is to be started.

Initialize Robot

4. On the left side of the footer, tap **Robot State** to initialize.

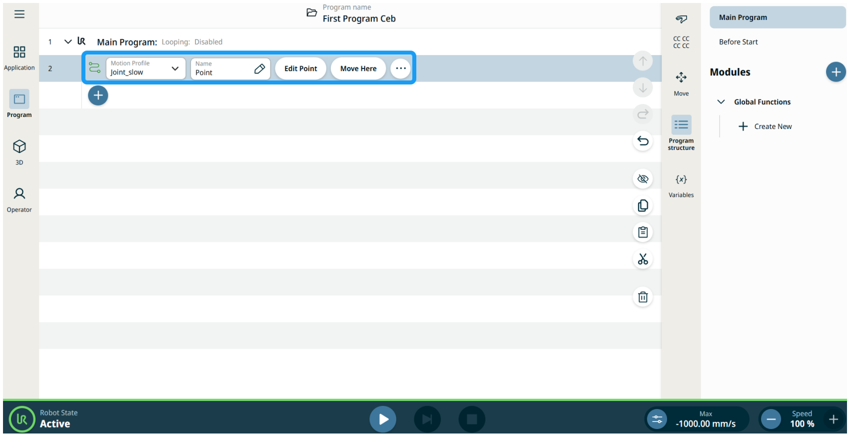
5. Tap **Power On**, then **Unlock**. Then you see the robot state as **Active**, and the activated button/slider of **Play**, **High Speed Manual Mode**, and **Speed**.



6. On the main navigation, tap the **Program** menu icon. See Command Nodes.

Move to Position A

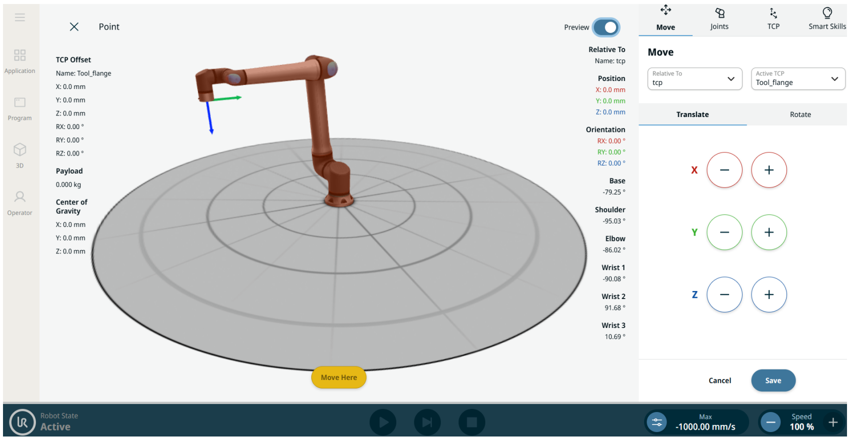
7. Tap Joint Move command. Note that in PolyScope X, the Waypoint node is merged to this command.



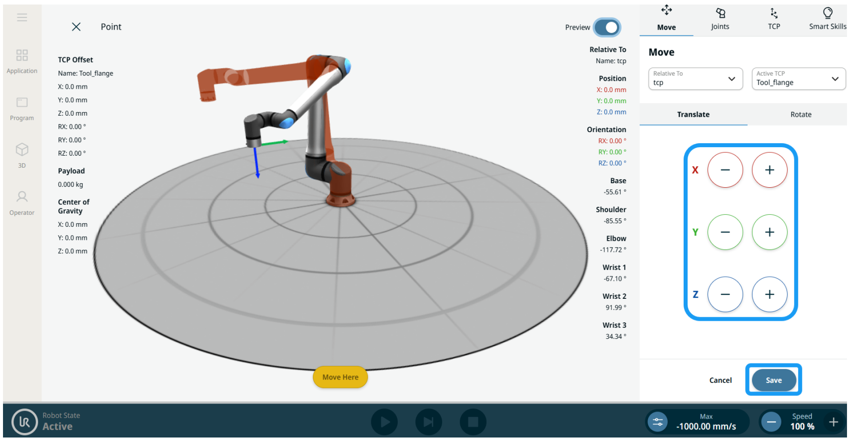
Two editable fields, two tabs, and an ellipses icon are inserted into the program tree:

- Motion Profile
- Name
- Edit Point
- Move Here

8. Tap the **Edit Point** tab. The screen with the robot arm in 3D appears. To the right of the screen, you see the Move menu.

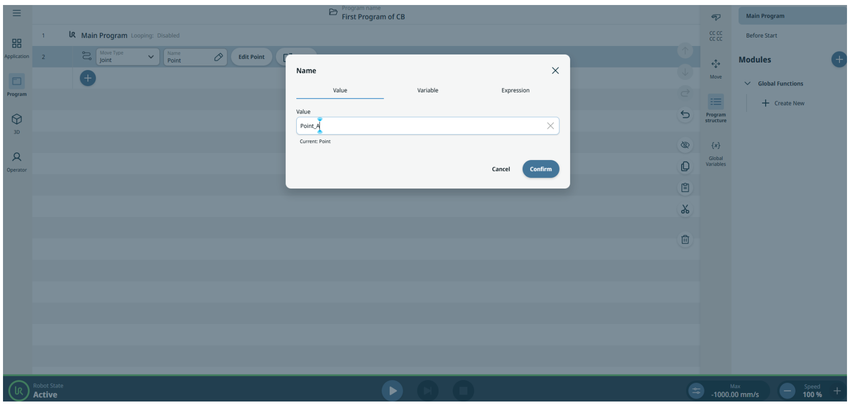


9. Use the **plus-minus button** of the X, Y, Z axes to jog the robot and set the position of your waypoint and **Save**.



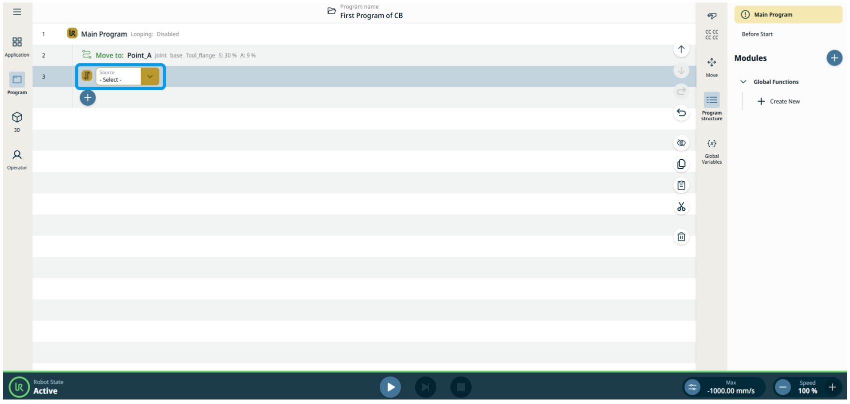
Name the first waypoint

10. In the **Name** field, enter **Point_A** as the name in the **Value** editor tab and **Confirm**.

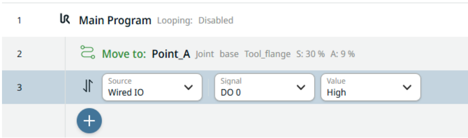


Set an IO

11. Tap the add icon in the program tree and the **Set** command icon. See Set.

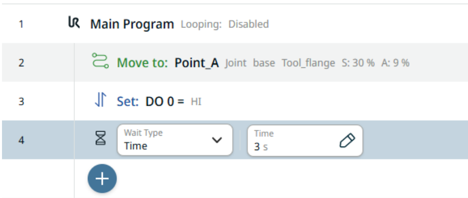


12. Tap the **Source** field and select **Wired IO**.
13. A **Signal** field is added. Select **DO 0**. A **Value** field is added to the right and select **High**.



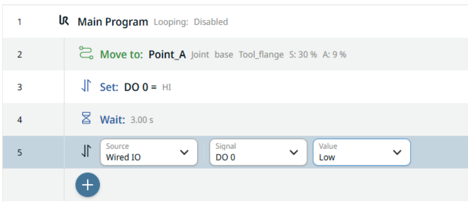
Insert Waiting time

14. Tap the add icon and choose **Wait** command. See Wait command.
15. In the **Wait Type** field, select **Time**.
16. Tap the **Time** field and enter **3** (three seconds) in the Value tab and **Confirm**.



Unset an IO

17. To create a second Set node, repeat **steps 11 to 13**.
18. In the **Signal** field, select **DO 0**. A Value field is added to the right and select **Low**.



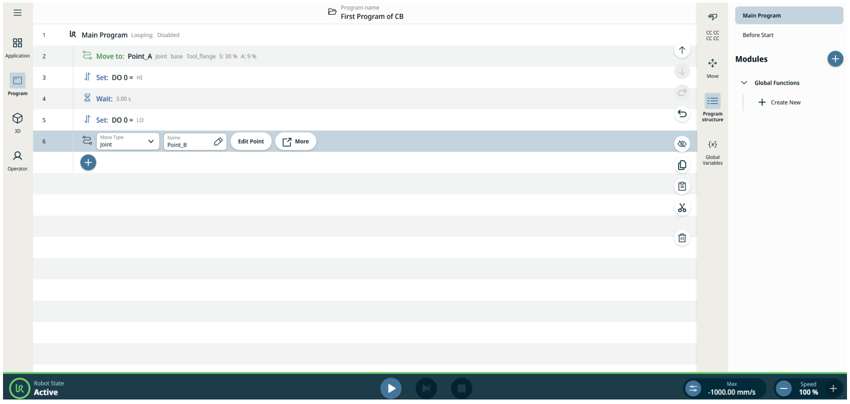
Move to Position B

19. Tap the add icon in the program tree. Repeat **steps 7 to 10** to create a second waypoint. Move the robot arm position in a direction clearly distinguished from Point_A.

Name the second waypoint

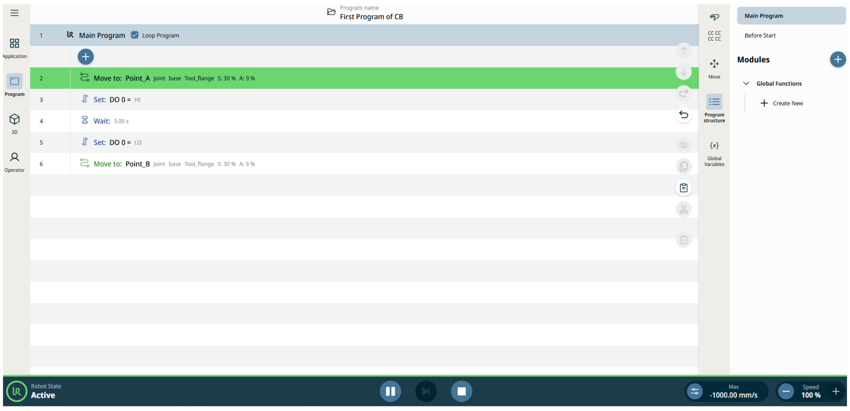
20. In the **Name** field, enter **Point_B** as the name in the **Value** editor tab and **Confirm**.

    The main screen shows you the first created program.

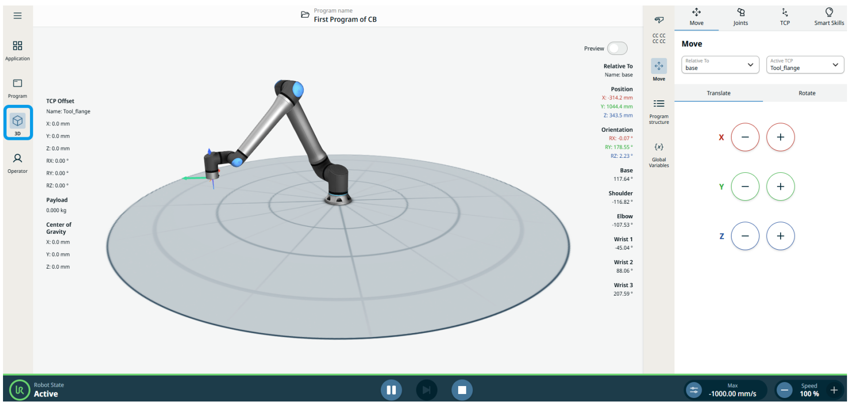


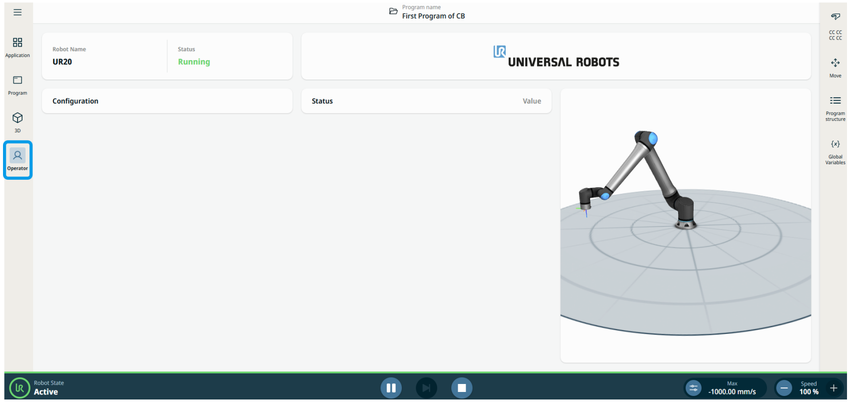
Verify program in 3D viewer and Communication application

21. Tap the **3D Viewer** icon on the main navigation.
22. On the footer, tap the **Play** button. When you are in the program tree screen, the program in green color signifies that it is the current program execution.

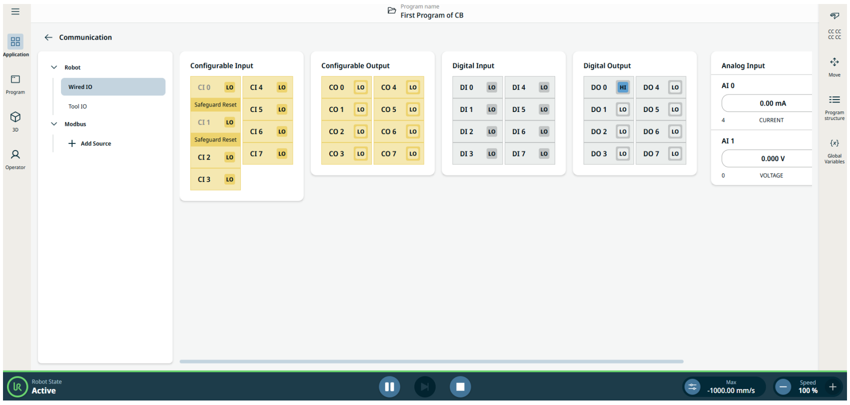


23. When you are in the 3D viewer and Operator screen, you see the robot move. On the side of the three-dimensional graphics, you can see the numerical details of the robot parts, position, and orientation.



24. 

25. Go to the **Application** tab and tap the **Communication** icon. See Communication. You see that **Digital Out 0 (DO 0)** updates (button turns blue) when you tap the play button in the footer.



WARNING

Keep your head and torso outside the reach (workspace) of the robot. Do not place fingers where they can be caught.

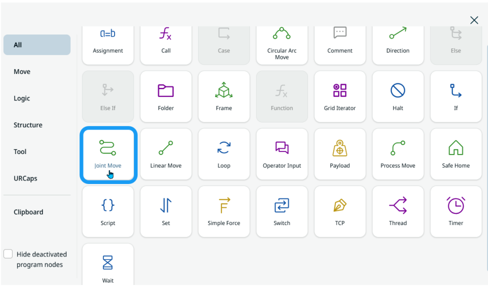
18. Pick and Place

Description

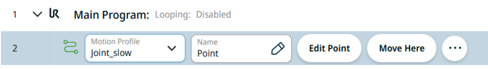
Pick and place is used when a robot picks up a piece from a tray or fixture and places it again in another tray or fixture. In a pick and place program, you create a default payload in the installation. Then you add a Set Payload when picking up an object. You update the payload after the gripper closes but before starting to move. Additionally, you use the Set Payload after the object has been released.

Creating a Pick-and-Place program

1. In the main navigation, tap the program icon. You are directed to the main program screen.
2. Tap the add icon to see the commands toolbox and choose the16.3.15 Joint Move on page 156 icon.



3. A command node with **Motion Profile** and **Name** fields, **Edit Point** and **Move Here** buttons, and ellipsis icon is inserted into the program tree.



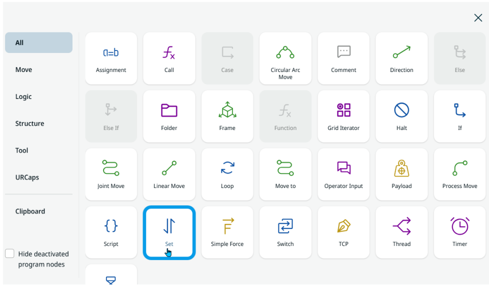
Position the robot above a tray/fixture

4. Tap **Edit Point** to mark the robot's first position above the corner of the tray or fixture. You are directed to the 3D viewer. Move the robot arm using the **plus-minus button** of the X, Y, Z coordinates and **Save**.
5. Name it as **Point_1** and tap **Confirm**.

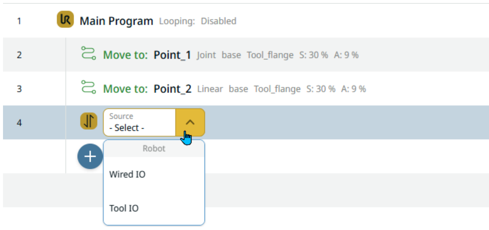
Robot grabs a thing from a tray/fixture

6. Tap the add icon and insert 16.3.16 Linear Move on page 163 command to grab the piece from the tray in a linear movement.
7. In the Edit Point, move the robot arm using the plus-minus button of the X, Y, Z coordinates , and Save. Jog down using the **Z axis** in blue arrow.
8. Name it as **Point_2**.

9. Tap the add icon. To close the gripper, choose the **Set** icon.

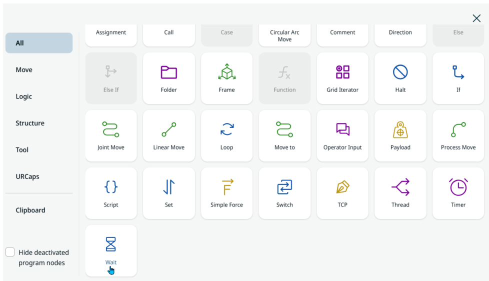


A command node with **Source** field is inserted into the program tree.

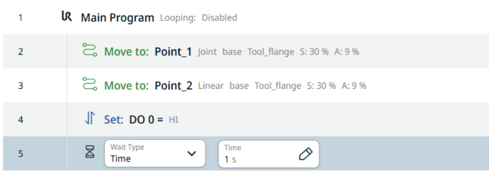


10. Choose **Tool IO** in the Source field. A Signal field is inserted to the right and choose **DO 0**. A Value field is further inserted and choose **High**.

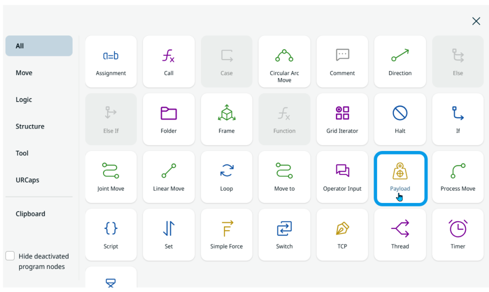
11. Tap back the add icon and insert a **Wait** command.



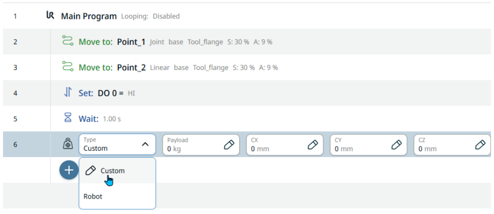
12. Choose **Time** in the Wait Type field and enter **1** in the Time field.



13. Tap back the add icon and insert a **Payload** command to update while the gripper is holding the piece.



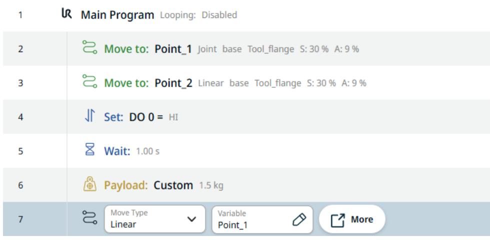
14. Choose Custom in the **Type** field. Four additional fields are added. In the **Payload** field, enter 1.5 kg in its Value tab, and in the **cz** field, enter 75 in its Value tab. Then **Confirm**.



Robot moves to the point above a tray/fixture

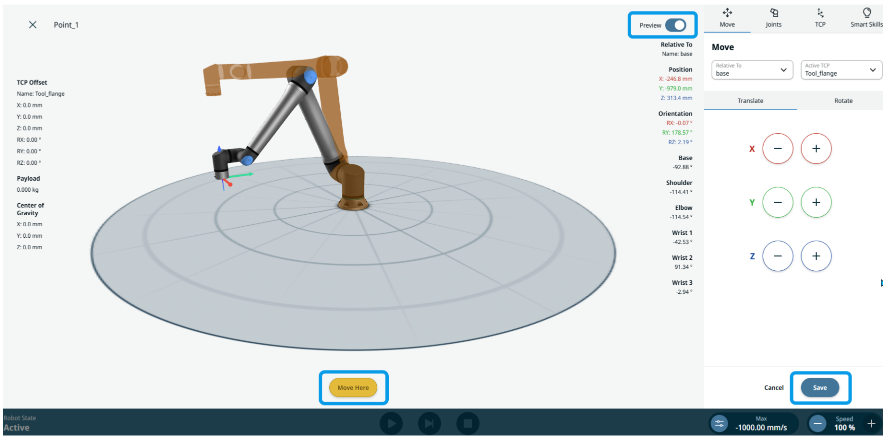
15. It is time to move up to the point above the tray again by using a variable.
16. Tap the add icon and insert a 16.3.16 Linear Move on page 163 command.
17. Tap the Name field, and in the Variable tab, select Point_1. Then Confirm.

    By using a variable position, you only have to reteach once, and not everywhere that position is used in case adjustments are needed.



Robot reaches the end of a tray/fixture

18. To move the robot into position, go back and tap the first16.3.15 Joint Move on page 156 command in the program tree.
19. Tap **Edit Point**, and you are directed to the 3D view of the robot. Press the **Move Here** button until the robot arm reaches the previewed position in brown color. The blue **Preview** button at the right top side turns to white, which indicates that you have reached the desired position. Tap Save.

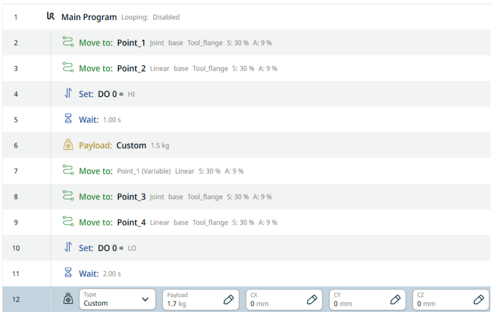


20. For the robot to reach the end of the tray, tap the last created command in the program tree.
21. Tap the add icon and insert another 16.3.15 Joint Move on page 156 command in the program tree.
22. In the Edit Point, move the robot arm using the plus-minus button of the X, Y, Z coordinates and Save. Jog sideways using the **X axis** in red arrow.
23. Name it as **Point_3**. Then Confirm.

Robot places a picked thing

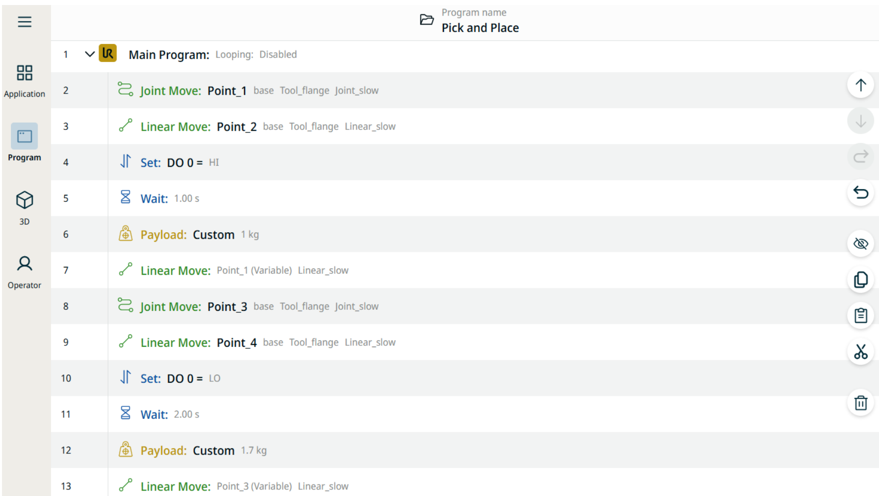
24. To get to the Place position, tap the add icon and insert another 16.3.16 Linear Move on page 163 command.
25. Tap Edit Point, and move the robot arm using the plus-minus button of the X, Y, Z coordinates and Save. Jog down using the **Z axis** in blue arrow.
26. Name it as **Point_4**. Then Confirm.
27. Tap the add icon and insert a **Set** command to open the gripper.
28. Select **Tool IO** in the Source field.
29. Select **DO 0** in the Signal field and **Low** in the Value field.
30. Insert another **Wait** command. In the Time field, enter 2 in the Value tab.
31. Tap the add icon and insert another Payload command while the gripper is empty.
32. Choose Custom in the Type field.

35. In the Payload field, enter 1.7 in the Value tab, and Confirm.



36. Insert another 16.3.16 Linear Move on page 163 command in the program tree to move the robot up to a point above the tray again by using a variable.
37. In the Name field, choose **Point_3** in the Variable tab, and Confirm.

    This is the final structure of your program tree.



38. Tap the **3D** Viewer in the main navigation and the **Play** button in the footer to view your created pick-and-place program.

19. Modules and Functions

Description

Modules and Functions is a feature of PolyScope X that will let you reuse functionality, increase readability, and make the program more maintainable, as you only have to adjust a single function to affect the entire program. Modules and functions are designed to be small and independent program blocks that make it easier to debug, reuse, and navigate.
The example that follows uses the created Pick and Place program.

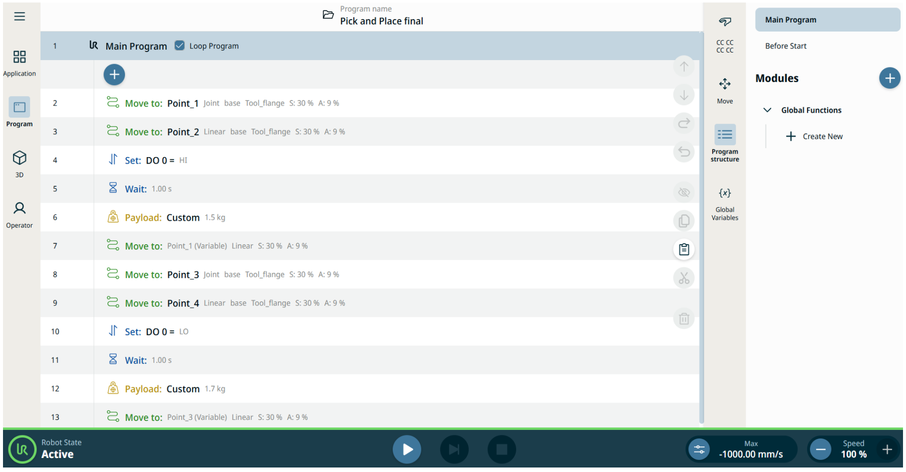
19.1. Gripper Module

Description

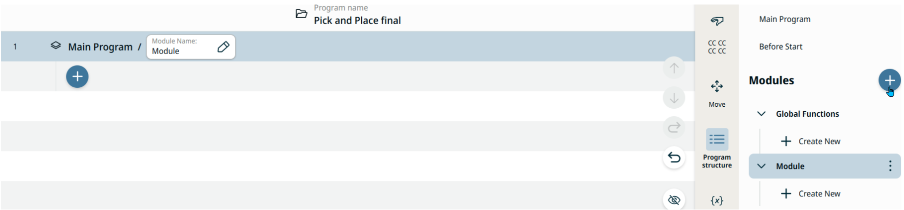
Grippers are devices that enable robots to pick up and hold objects. When combined with the robot arm, grippers enable you to automate processes, such as inspection, assembly, pick and place, and machine tending. Similar to the human hand, grippers are positioned at the end of the arm, functioning as end effector, and their abilities allow you to combine the strength of an arm with the dexterity of a hand. When you create a gripper module, you optimize ease, readability, and flexibility in using its appropriate command nodes.

To create Gripper module

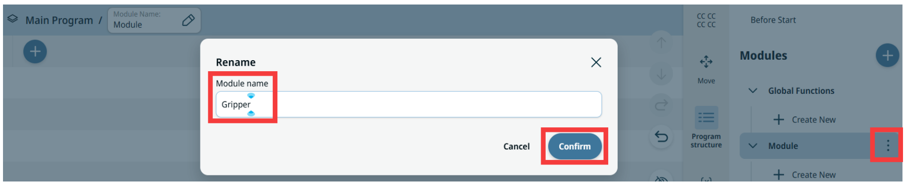
1. Open the created pick-and-place program in the System Manager. In the main screen, you see all the used commands in the program tree.



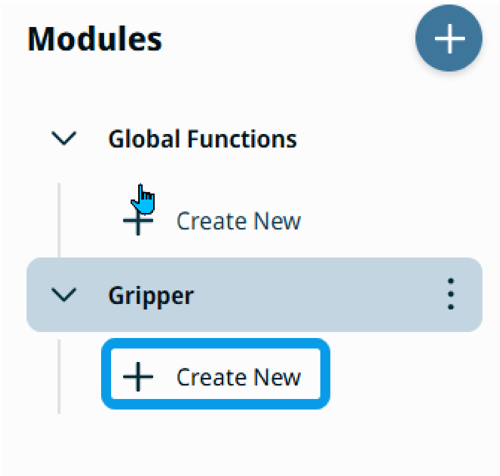
2. Make a new module which will contain the functions for Gripper. Tap the add icon of the modules tree in the multitask screen. A module is inserted below Global Functions, and the main screen is depopulated of the created commands.



3. Tap the Module kebab icon, rename the module name as Gripper, and Confirm.



4. Below the created Gripper module, tap + Create New.

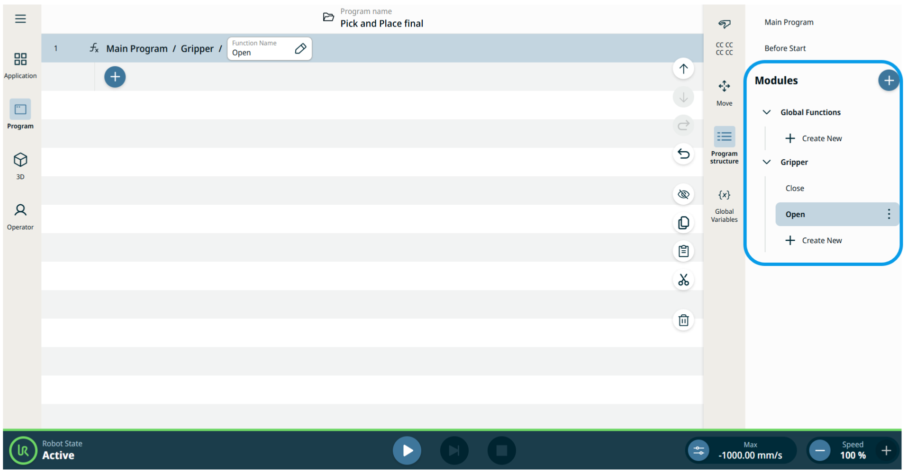


5. In the main screen, tap the Function Name field in the Main Program tree, create name as Close, and Confirm.

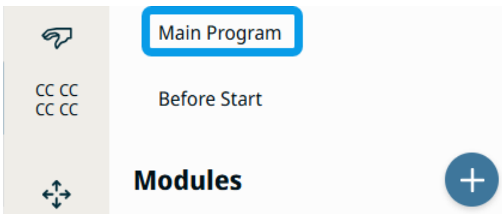
6. In the multitask screen, tap + Create New below the newly created Close function of Gripper.

7. In the main screen, tap the Function Name field in the Main Program tree, name it as Open, and Confirm.

   This is how PolyScope X interface appears in your screen. Take particular notice on the created Modules tree in the multitask screen.



8. Tap Main Program in the topmost part of the multitask screen. All used commands become visible again in the main screen.



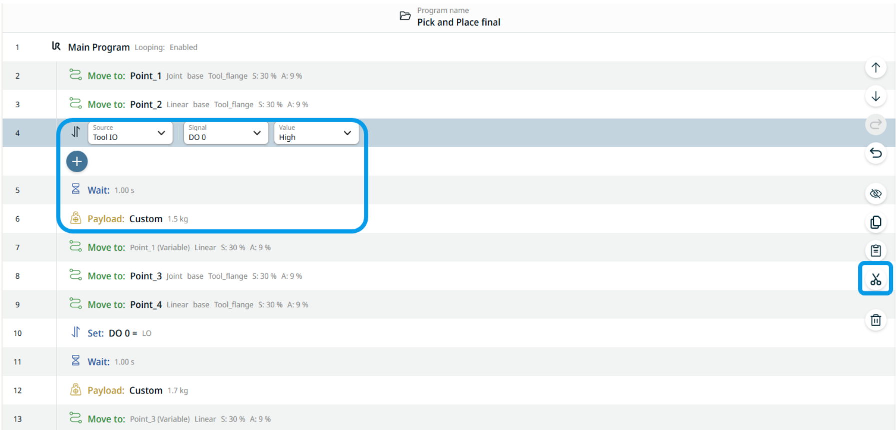
19.1.1. Gripper Close Function

Description

A gripper close function ensures that right timing, precise force, and correct payload are applied.

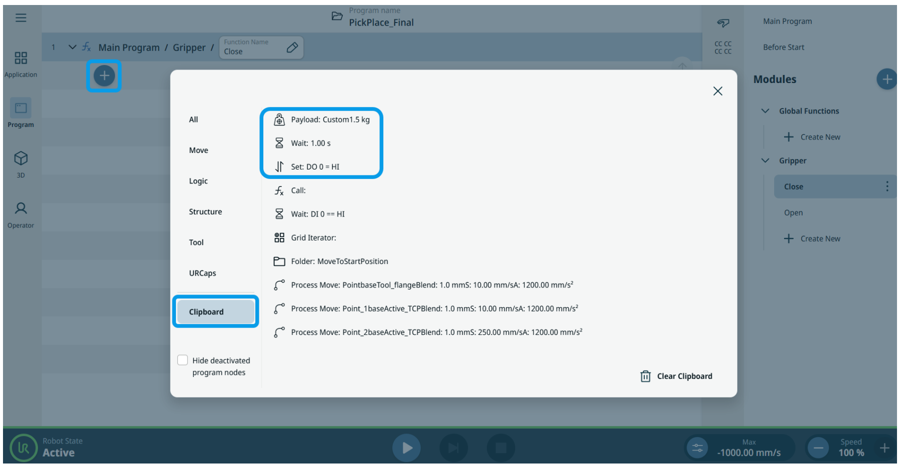
To create Close Gripper Function

1. Cut out the first set of Set, Wait, Payload commands.

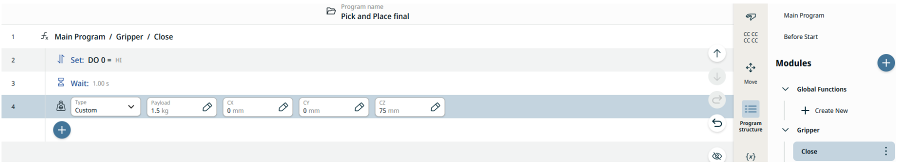


2. Tap the Set command node and tap the cut icon found on the right side of the main screen.
3. Tap the Wait command node and tap the cut icon.
4. Tap the Payload command and tap the cut icon.

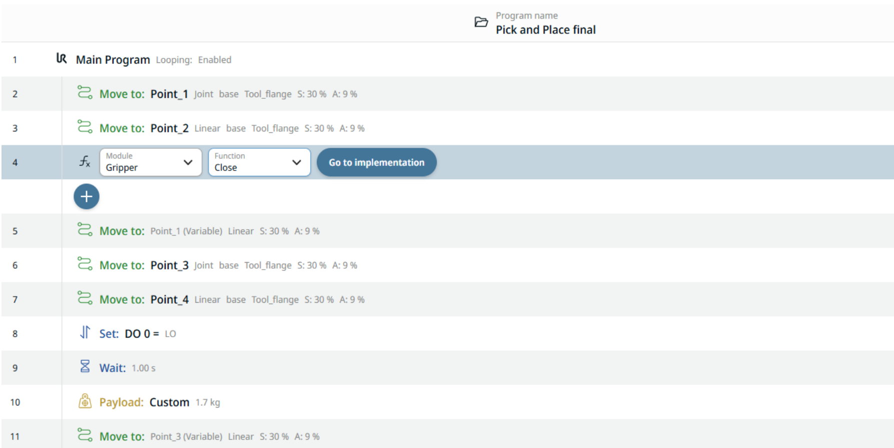
   These three commands are transferred in the clipboard toolbox.
5. In the multitask screen, tap the Close function of Gripper.
6. Tap the add icon in the main screen and choose the Clipboard toolbox.



7. In sequential order, insert the Set, Wait, Payload commands into the Close Gripper function.
   7.1. Tap the Set command.
   7.2. Tap the add icon below.
   7.3. Tap the Wait command.
   7.4. Tap the add icon below.
   7.5. Tap the Payload command.



8. Tap again the Main Program to insert the Gripper Close Function into the program tree.
9. Tap the command node **Move to: Point_2** and then the add icon.
10. Choose the Commands toolbox. Tap the Call icon, and a node with the Module and Function fields appear.
11. Select Gripper in the Module field and Close in the Function field.



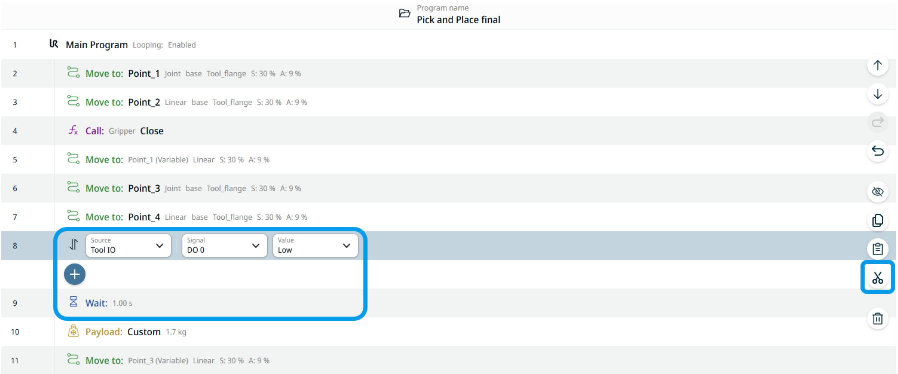
You are able to program a Gripper Close Function once, and whenever you need to close the gripper, just call the function.

19.1.2. Gripper Open Function

Description	A gripper open function ensures that right timing to open the gripper is applied.

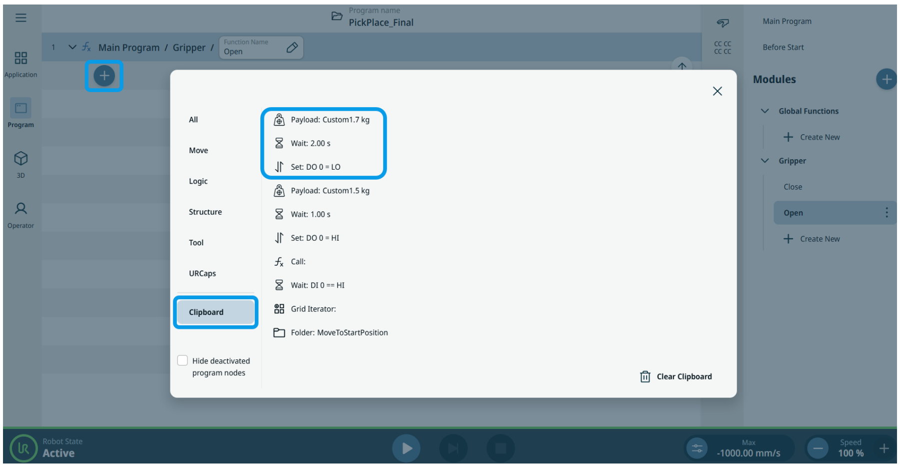
To create Open Gripper Function

1. Cut out the second set of Set, Wait, Payload commands.

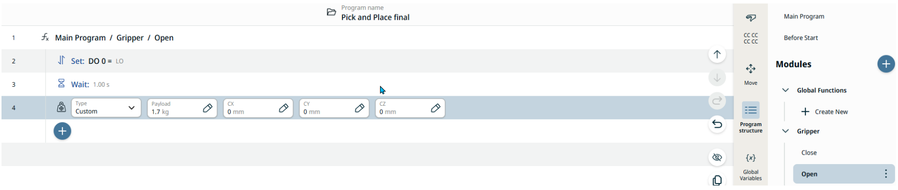


2. Tap the Set command node and tap the cut icon found on the right side of the main screen.
3. Tap the Wait command node and tap the cut icon.
4. Tap the Payload command and tap the cut icon.

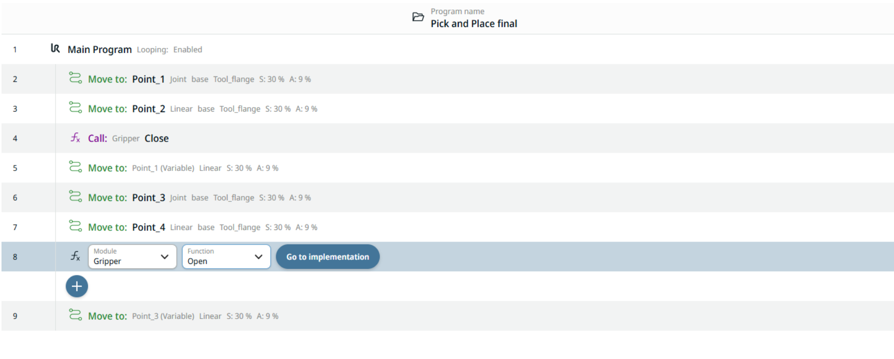
   These three commands are transferred in the clipboard toolbox.
5. In the multitask screen, tap the Open function of Gripper.
6. Tap the add icon in the main screen and choose the Clipboard toolbox.



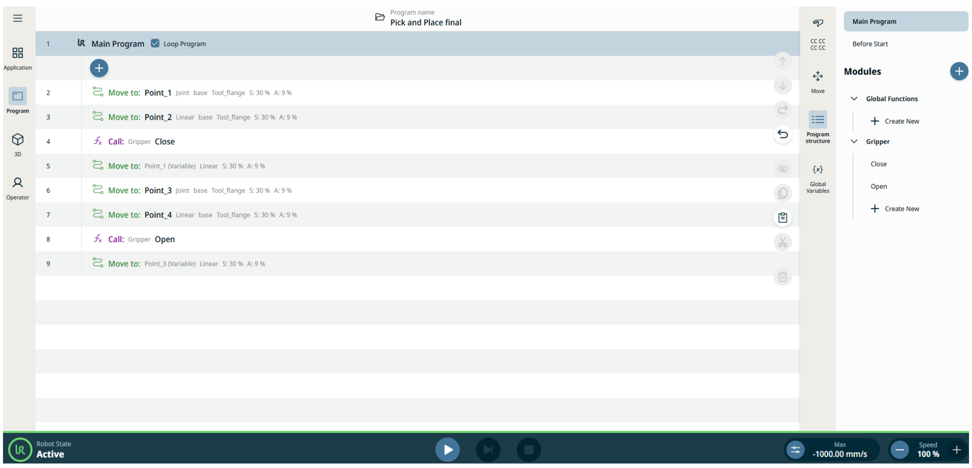
7. In sequential order, insert the second Set, Wait, Payload commands into the Open Gripper function.
   7.1. Tap the Set command.
   7.2. Tap the add icon below.
   7.3. Tap the Wait command.
   7.4. Tap the add icon below.
   7.5. Tap the Payload command.



8. Tap again the Main Program to insert the Gripper Open Function into the program tree.
9. Tap the command node **Move to: Point_4** and then the add icon.
10. Choose the Commands toolbox. Tap the Call icon, and a node with the Module and Function fields appear.
11. Select Gripper in the Module field and Open in the Function field.



You are able to program a Gripper Open Function once, and whenever you need to open the gripper, just call the function.
This is now how your program tree appears with the applied module and functions, but with the same robot movement as before:



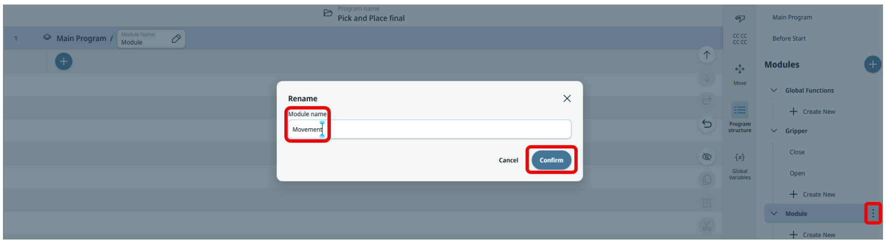
19.2. Movement Module

Description

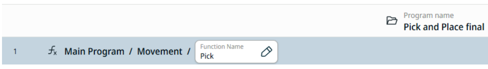
A movement module enables you to reuse Move To command nodes to pick and place an object from one place to another using a gripper.

To create Movement Module

1. Tap the add icon of the modules tree in the multitask screen. A module is inserted below Gripper module.
2. Tap the Module kebab icon, rename the module name as Movement, and Confirm.

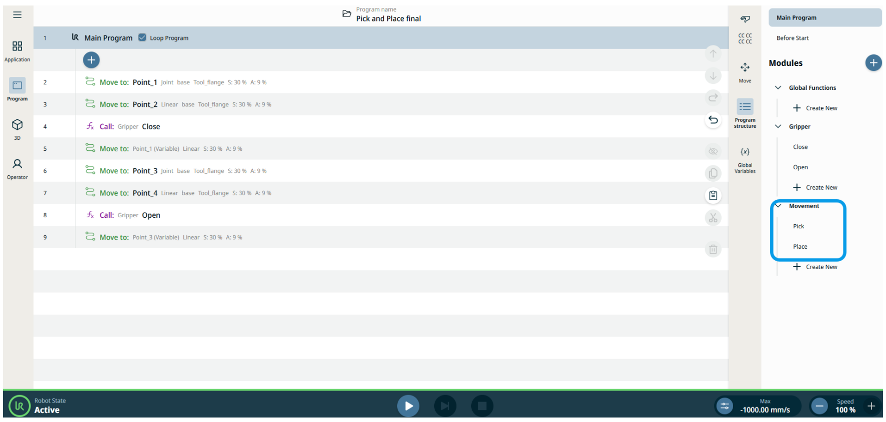


3. Below the created Movement module, tap + Create New.
4. In the main screen, tap the Function Name field in the Main Program tree, create name as Pick, and Confirm.



5. In the multitask screen, tap + Create New below the newly created Pick function of Movement.
6. In the main screen, tap the Function Name field in the Main Program tree, name it as Place, and Confirm.
7. Tap Main Program in the topmost part of the multitask screen. All used commands become visible again in the main screen.

   This is how PolyScope X interface appears in your screen.



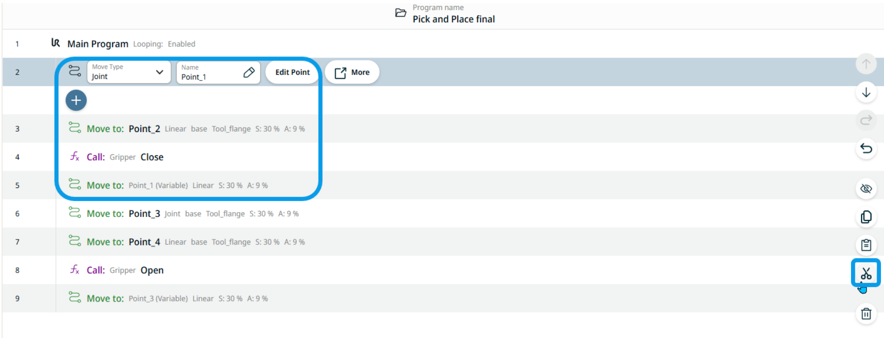
19.2.1. Pick Movement Function

Description

A pick function uses the move command in the direction of the object to be picked and the call command to close the gripper.

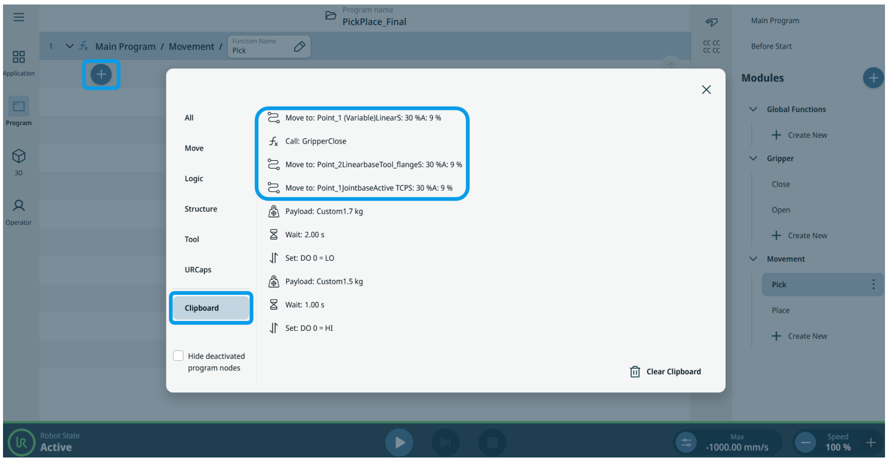
To create Pick Movement Function

1. Cut out the first set of Move to commands, including the Call command for gripping.

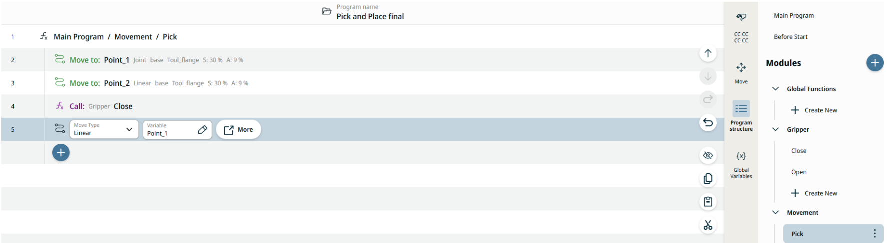


2. In alternate order, tap each command nodes and the cut icon one after the other.

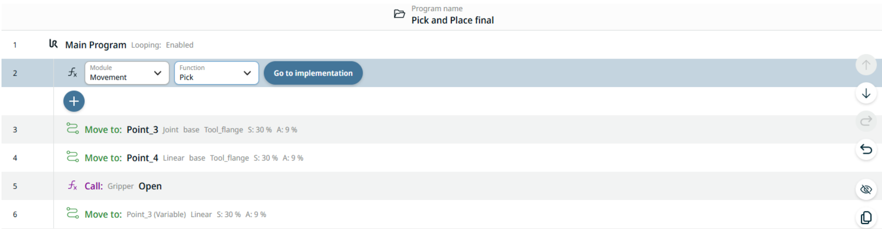
   The four commands are transferred in the clipboard toolbox.
3. In the multitask screen, tap the Pick function of Movement.
4. Tap the add icon in the main screen and choose the Clipboard toolbox.



5. In sequential order, insert the first set of commands into the Pick Movement function.
   5.1. Tap the Move to: Point_1 command.
   5.2. Tap the add icon below.
   5.3. Tap the Move to: Point_2 command.
   5.4. Tap the add icon below.
   5.5. Tap the Gripper Close function.
   5.6. Tap the add icon below.
   5.7. Tap the Move to: Point_1 (Variable) command.



6. Tap again the Main Program to insert the Movement Pick Function into the program tree.
7. Tap the add icon, choose the Commands toolbox, and tap the Call icon. A node with the Module and Function fields appear.
8. Select Movement in the Module field and Pick in the Function field.



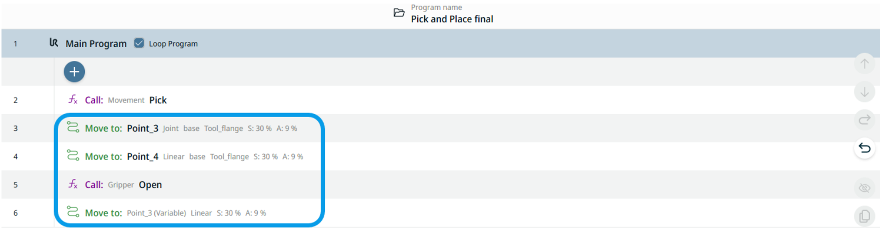
19.2.2. Place Movement Function

Description

A place function uses the move command in the direction of the place where object is to be put and the call command to open the gripper.

To create Place Movement Function

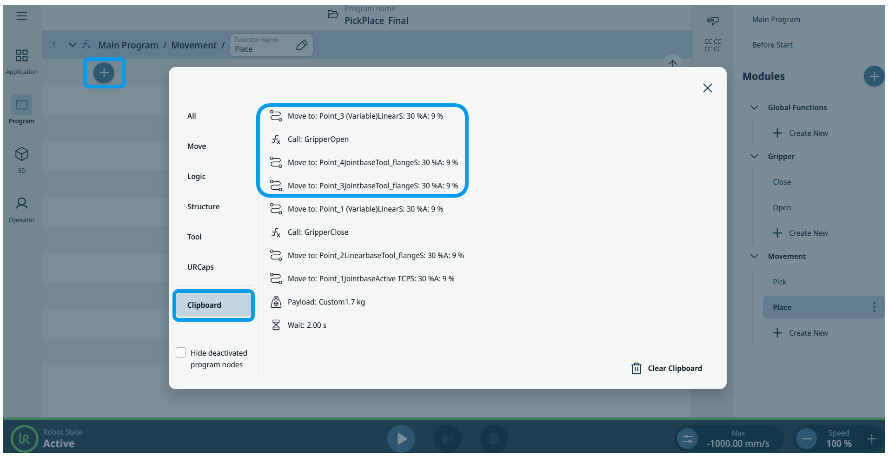
1. Cut out the second set of Move to commands, including the Call command for gripping.



2. In alternate order, tap each command nodes and the cut icon one after the other.

   These four commands are transferred in the clipboard toolbox.

3. In the multitask screen, tap the Place function of Movement.
4. Tap the add icon in the main screen and choose the Clipboard toolbox.



5. In sequential order, insert the first set of commands into the Place Movement function.

   5.1. Tap the Move to: Point_3 command.

   5.2. Tap the add icon below.

   5.3. Tap the Move to: Point_4 command.

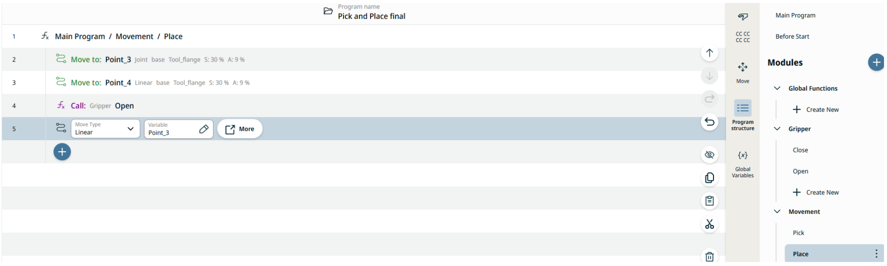
   5.4. Tap the add icon below.

   5.5. Tap the Gripper Open function.

   5.5. Tap the add icon below.

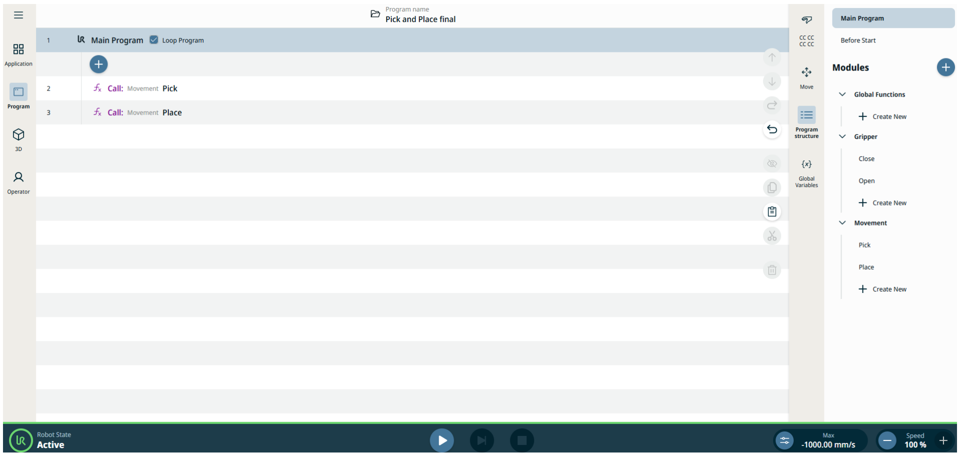
   5.6. Tap the Move to: Point_3 (Variable) command.

6. Tap again the Main Program to insert the Movement Place Function into the program tree.



7. Tap the command node **Call: Movement Pick** and then the add icon.
8. Choose the Commands toolbox. Tap the Call icon, and a node with the Module and Function fields appear.
9. Select Movement in the Module field and Place in the Function field.

You can see the program tree of the PolyScope X interface when structured in functions and modules:



In structuring your program by functions and modules, it is possible to call functions within functions to make everything even more reusable. Creating a full program mainly comprised of functions will make it easier for you to manage long programs. Furthermore, you make the program more maintainable and avoid repetition in your programing. As you can see on the 3D viewer and Operator screen, the robot movements are still the same as before.

20. Application Tab

The Application tab allows you to configure the settings which affect the overall performance of the robot and PolyScope X.

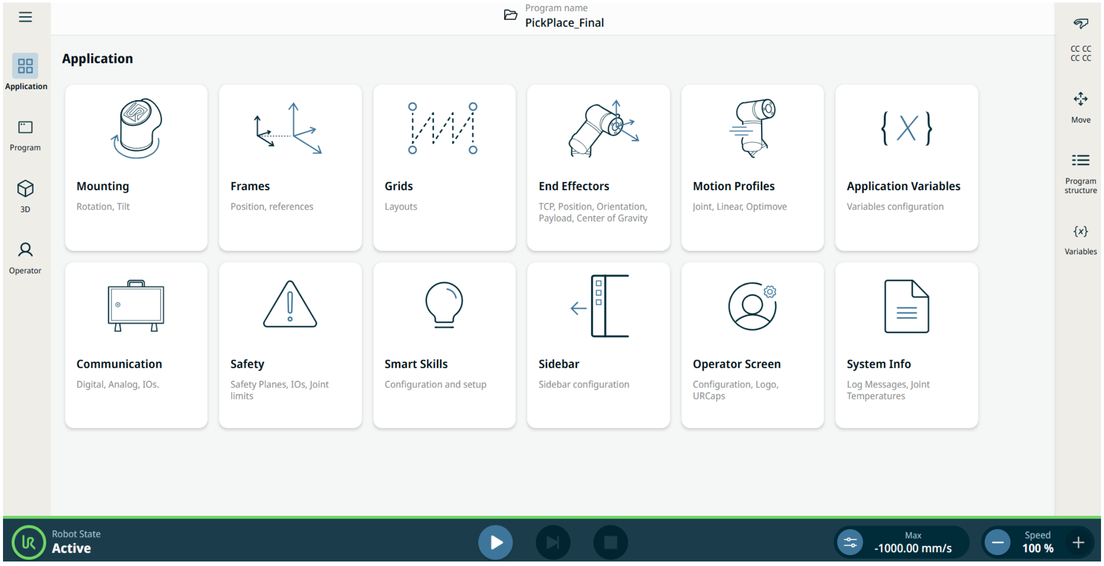


Figure 1.2: *Application screen displaying application buttons.*

Use the Application tab to access to the following configuration screens:

- Mounting
- Frames
- Grids Application
- End Effectors Application
- Motion Profiles
- Application Variables
- 20.7 Communication on page 244
- Safety
- 20.9 Smart Skills Application on page 294
- 20.10 Sidebar in Application Tab on page 298
- 20.11 Operator Screen Application on page 301
- 20.12 System Info on page 302

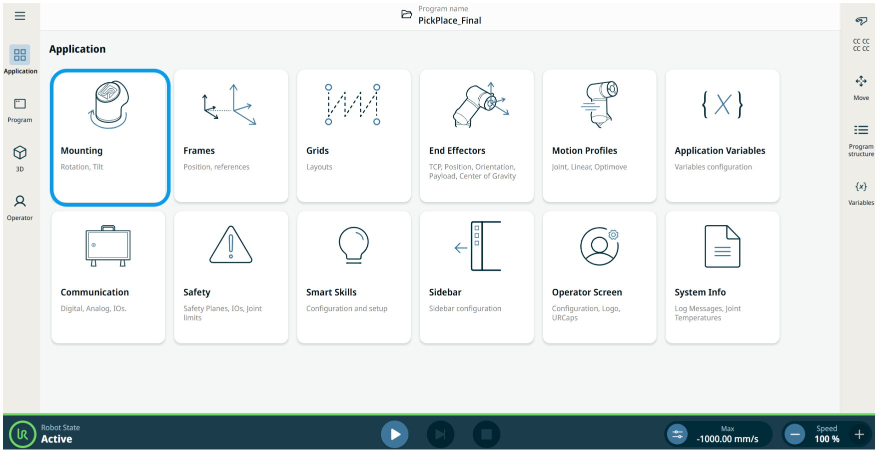
20.1. Mounting

Description

Mounting application contains settings to determine if the robot is mounted vertically, horizontally, or at an angle. It allows you to configure the orientation of the robot base, to determine the direction of gravity, to enable smooth and precise movement, and to give the robot accurate appearance on the visualizations.

Using the Mounting application functionality

1. Go to the application nodes screen. See Application Tab.
2. Tap the **Mounting** icon.

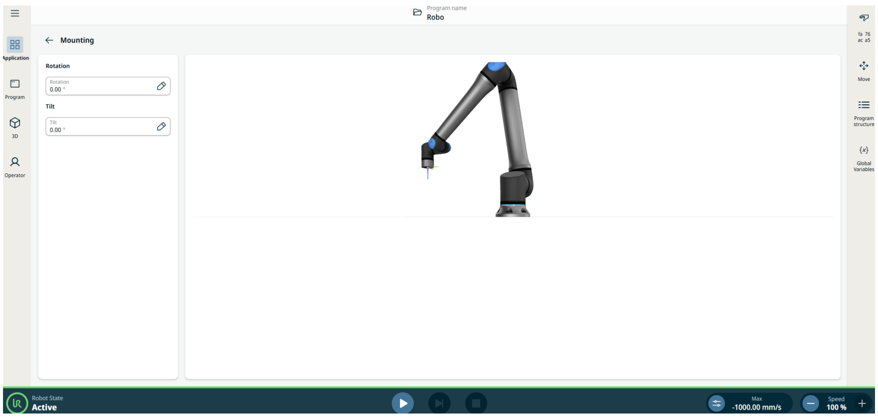


A screen appears with two fields on the left side:

Rotation. The alignment with respect to the power cable.

Tilt. The angle of the base in degrees: 0 being horizontal floor mounting, 90 being wall mounting, and 180 being a ceiling mounting.

The robot arm is seen in the center screen.

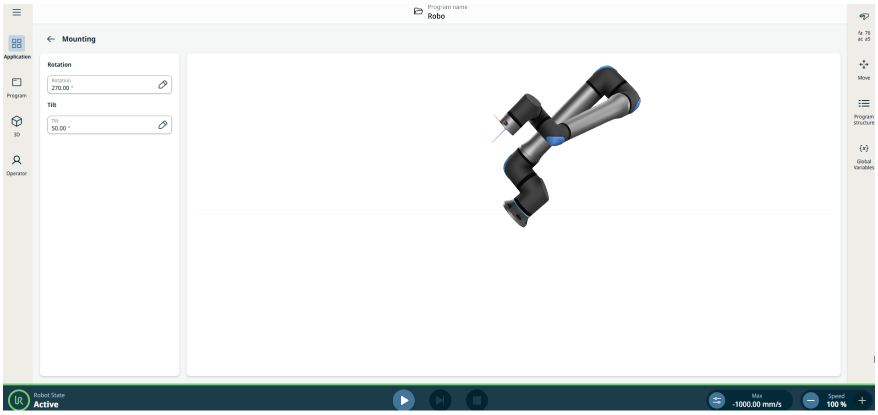


3. Use the following touch gestures for the robot arm.
    - **Press and rotate** clockwise or counterclockwise.

      Function: Changes your viewing direction of the robot.
    - **Pinch**. Touch the screen with two fingers and bring them closer together.

      Function: Zooms out the robot image.
    - **Spread**. Touch the screen with two fingers and move them apart.

      Function: Zooms in the robot image.
4. Tap the **Rotation** field, enter your value in degrees, and **Confirm**.
5. Tap the **Tilt** field, enter your value in degrees, and **Confirm**. The corresponding change in the robot arm is seen.



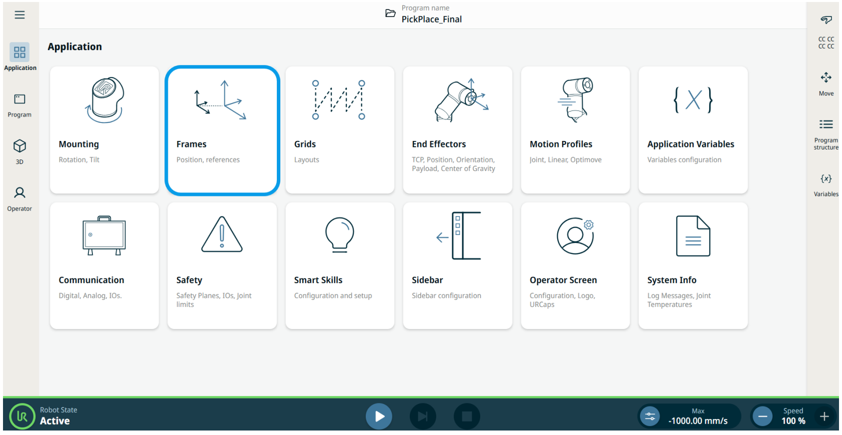
20.2. Frames

Description

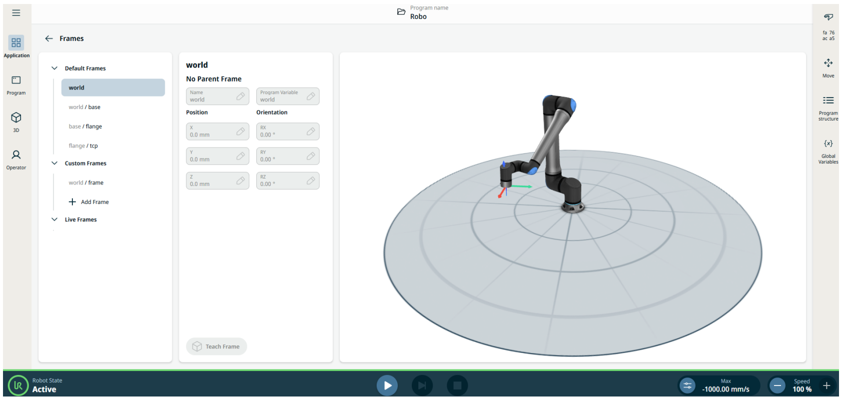
Frames application allows you to define frames of reference for use in a program.

Using the Frames application functionality

1. Go to the application nodes screen. See Application Tab.
2. Tap the **Frames** icon.



A screen appears, which is divided into three panels.



The left panel contains the **Default Frames**, **Custom Frames**, and **Live Frames**.

The center panel contains eight fields:

- Name field
- Program Variable field
- X Position field
- Y Position field
- Z Position field
- RX Orientation field
- RY Orientation field
- RZ Orientation field

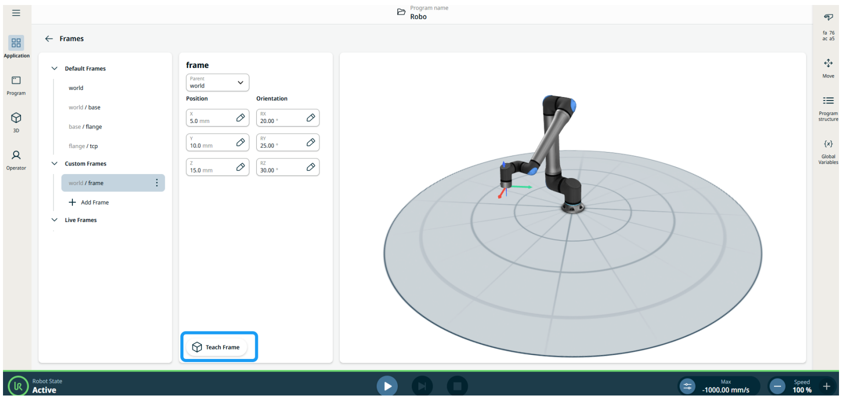
The right column/panel shows the robot arm in X, Y, Z coordinates.

Under the **Default Frames** are found the three predefined PolyScope X frames:

- Base. This is the center of the base of the robot. If your robot is mounted in a fixed location, this frame will never need to change, and other fixed locations can be defined relative to it.
- TCP. This is the position of the active TCP. It will update as the robot moves when jogged or while a program is running.
- World. When the robot is mounted in a fixed location, this frame is the same as the base. However, if your robot is mounted on a moveable rail or gantry, it is then possible to update the base frame as the robot moves, while the world frame is always fixed.

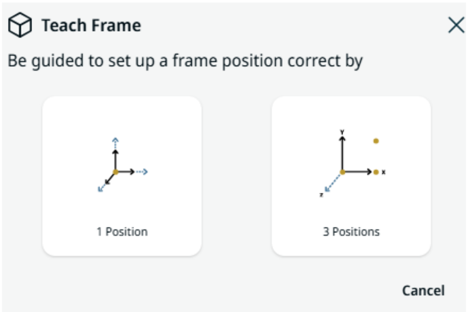
  Selecting any of the predefined frames will allow you to view their values. However, they cannot be edited.

3. Under the **Custom Frames**, tap **+ Add frame**. The eight fields in the center column/panel are activated.
4. In the **Parent** field, choose your frame.
5. Edit your value in the **X, Y, Z Position** fields and **Confirm**.
6. Edit your value in the **RX, RY, RZ Orientation** fields and **Confirm**.
7. Tap **Teach Frame** button found at the bottom of the panel.

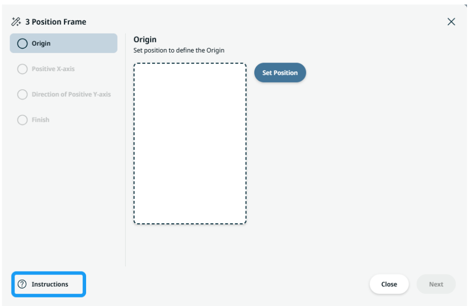


A pop-up screen appears with two options to set up a frame position.

- 1 Position
- 3 Positions

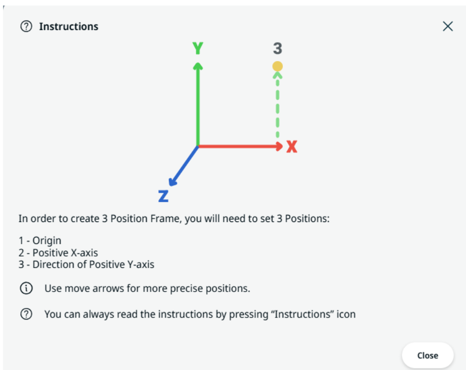


8. When you select **1 Position**, the 3D viewer becomes the main screen. Use the plus-minus button to translate and rotate the robot arm's three coordinates.
9. Tap **Save**.
10. When you select **3 Positions**, a pop-up screen appears. Tap the **Instructions** icon on the bottom left side.



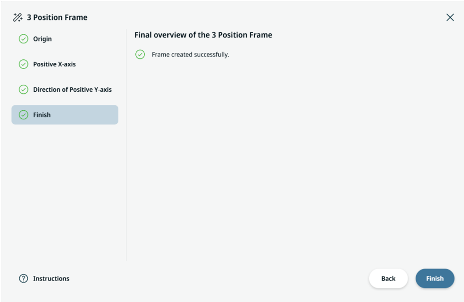
The Instructions box on how to create 3-Position Frame appears.

11. Tap **Set Position** for **Origin**.



12. The 3D viewer becomes the main screen. Use the plus-minus button to translate and rotate the robot arm's three coordinates.
13. Tap **Save**.
14. An option to edit position is provided. If there is no need to edit, tap **Next**.
15. Tap **Set Position** for **Positive X-axis**.
16. Repeat steps 12-14.

17. Tap **Set Position** for **Direction of Positive Y-axis**.
18. Repeat steps 12–14.
19. Tap **Finish** when frame is created successfully.



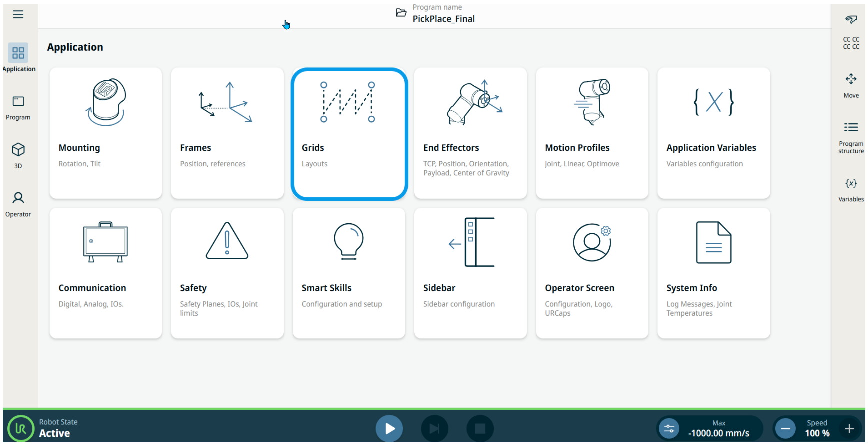
20.3. Grids Application

Description

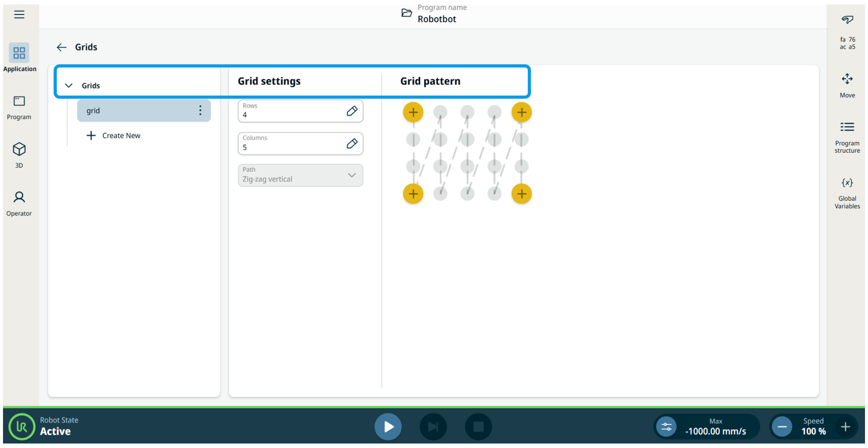
The **Grids** application is used to set up a grid, such as for palletizing, where you define four corners and the number of positions between them. This is like the palletizing wizard in PolyScope 5.

Using the Grids application functionality

1. Go to the Application menu screen. See Application Tab.
2. Tap the **Grids** icon.

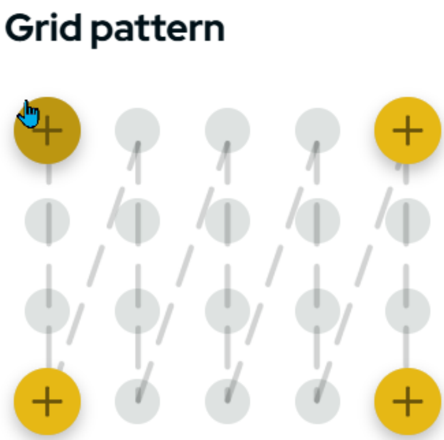


In the main screen, you can see the grids tree on the left panel, and the grid settings and grid pattern on the right panel.



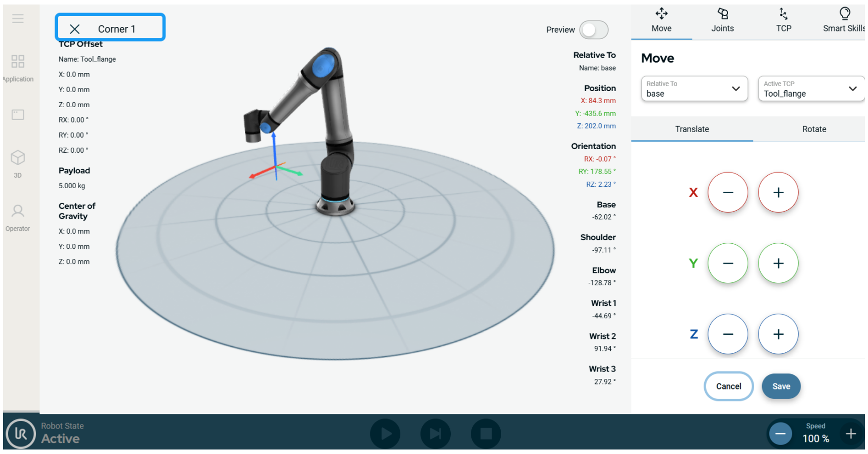
3. Tap **Create New** on the grids tree.
4. Enter the grid name in the **create new grid** field and **Save**.
5. Enter the number of **rows** and **columns** in the fields provided under the grid settings and tap **Confirm**. The grid pattern corresponds with what you input in the grid settings.

6. In the grid pattern, tap the first corner (in yellow circle).



   You will be directed to the 3D viewer of the robot arm in X, Y, Z coordinates.

7. On the right side of the screen, tap the **minus-plus button** in each coordinates to translate and rotate. Tap **Save**.



8. Press and rotate, pinch and spread the 3D screen to zoom in or zoom out the robot parts, take a close-up view, rotate sideways, and other viewing direction.
9. Repeat steps 8-11 thrice to make a grid pattern for the remaining corners 2-4.

When your Grid is set up, you can implement it in your program.

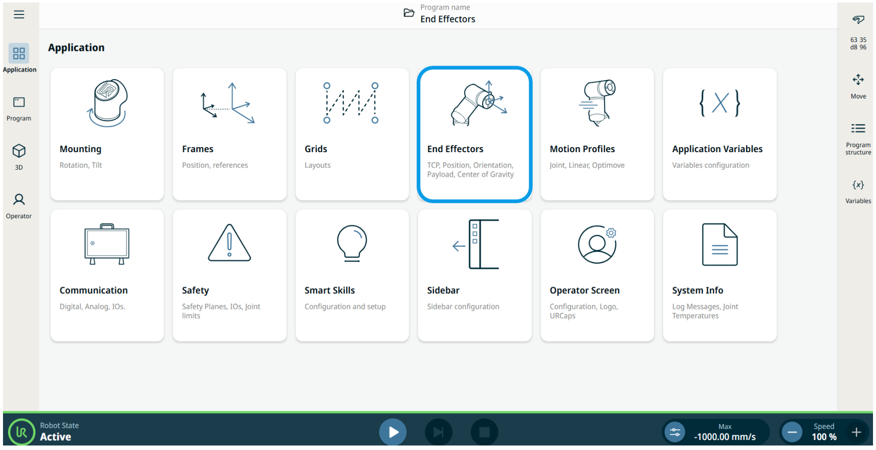
20.4. End Effectors Application

Description

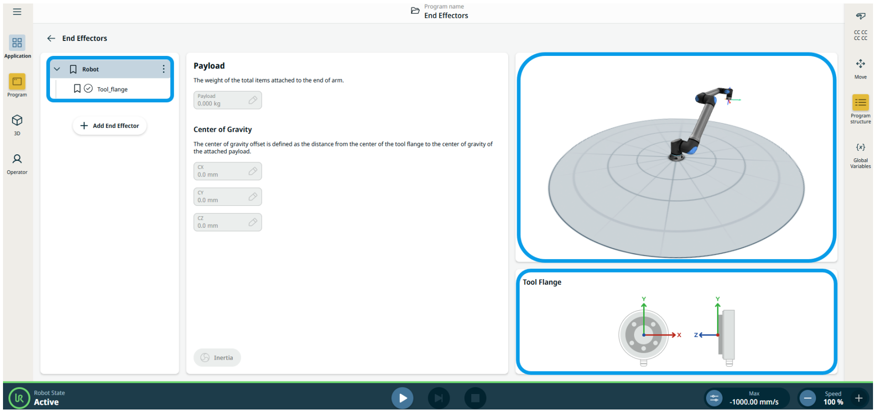
The End Effectors application is used for configuring the tooling at the end of the arm, such as the tool center point (TCP) and center of gravity (CoG).

Using the End Effectors application functionality

1. Go to the Application nodes screen. See Application Tab.
2. Tap the **End Effectors** icon.



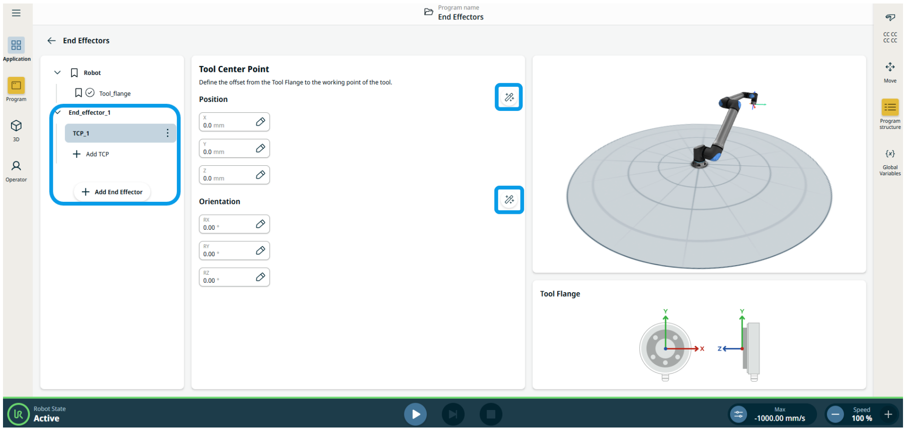
In the main screen, you can see End Effector application screen with a 3D robot view and an illustration of the Tool flange on the right panel. This change has been introduced to help users visualize the frame at the TCP and Tool Flange. On the left panel, the end effectors list now has the **Robot** group. The built-in **Tool_flange** end effector is located under that group.



Tool Center Point (TCP) wizards

The TCP wizards enables you to define the position and orientation of a TCP. The new wizards are accessible from the Tool Center Point application screen by tapping the new wizard icon next to the **Position** header and the **Orientation** header, respectively. In both wizards, you can call up a popup any time with more detailed instructions and animations on how to use the wizard. The instructions can be accessed by pressing the **Instructions** icon in the bottom left corner of the wizards.

1. On the left panel, tap **+ Add End Effector**.
2. Tap **TCP_1** under the newly added **End_effector_1 tree**.
3. In the central panel, the Tool Center Point, you see the two wizard wand icons for Position and Orientation.



Adding, Renaming, Duplicating, and Deleting TCP

You can start configuring a new TCP with the following actions:

- Tap + to define a new TCP with a unique name. The new payload appears in the End effector tree..
- Tap to rename a TCP.
- Tap to duplicate a TCP.
- Tap to delete a selected TCP. You cannot remove the last TCP.

Active TCP

The check-mark in the drop-down indicates which TCP is active TCP_1.

Default TCP

The default TCP is set as the active TCP before the program starts.
Select the desired TCP and tap to **Set as default** to set a TCP as the default.

20.4.1. TCP Position

Description

The TCP position wizard assists user in finding the position (translation) of a TCP. This is useful in situations where the position is unknown and cannot be easily measured.

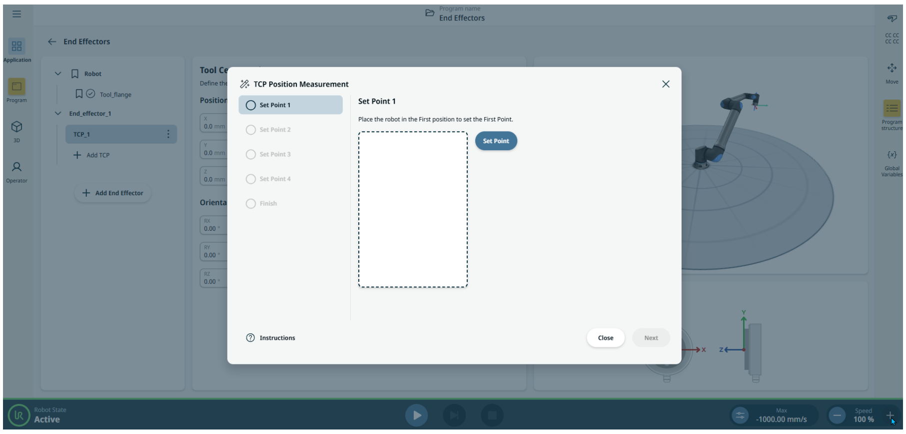
TCP Position wizard

1. On the central panel, tap the TCP Position wizard icon,

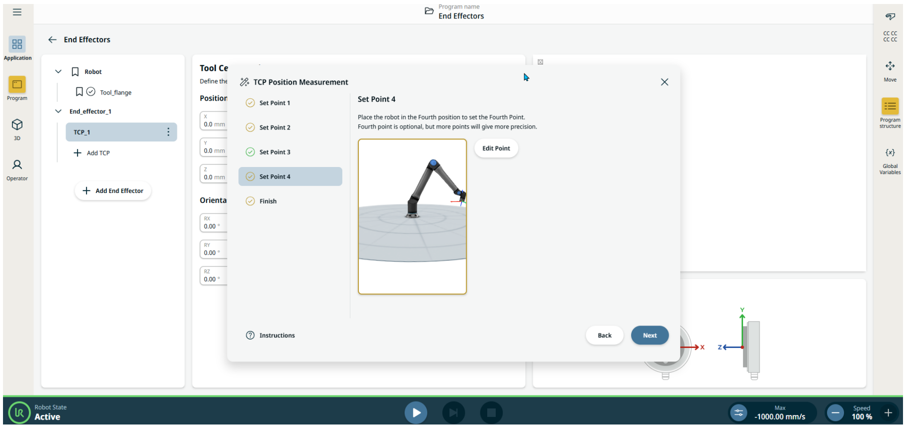
   Position

   and the pop-up TCP Position Measurement screen appears.

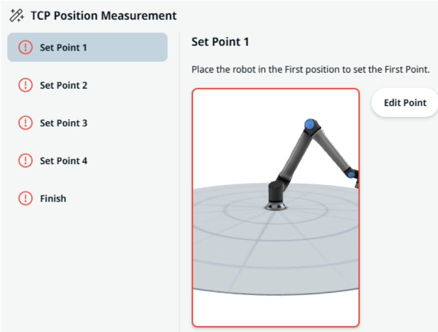
2. Tap the **Set Point** button to set three or four positions by moving the TCP to the same position from different angles. This can be done by freedrive or jogging the robot in the 3D interface. You need at least three points from different angles for successful measurement.

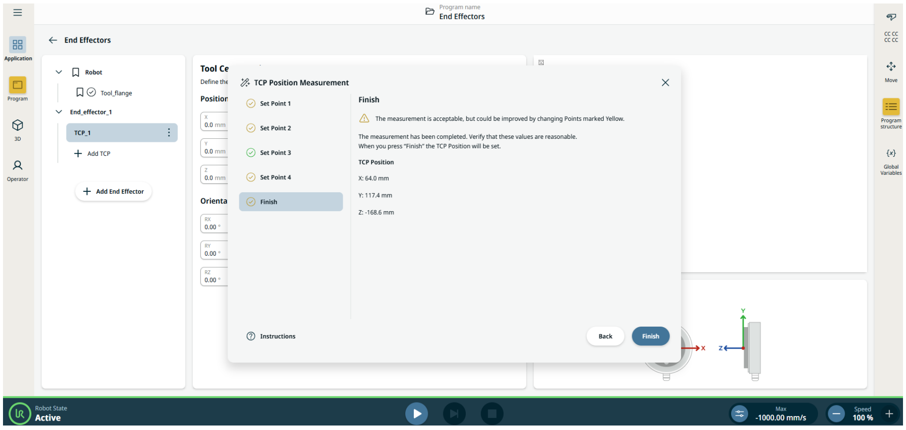
   The wizard calculates the TCP position automatically. Though three positions are sufficient to determine the TCP position, a fourth position can be used to further verify that calculations are correct.

NOTICE

A red warning in Set Points informs user if the taught positions are not sufficiently diverse, in which case it is not possible to compute the TCP position. This means that the positions must be retaught.



3. When the measurement is completed, tap **Finish**. You can see the robot arm's position as shown in the X, Y, Z coordinates.



20.4.2. TCP Orientation

Description

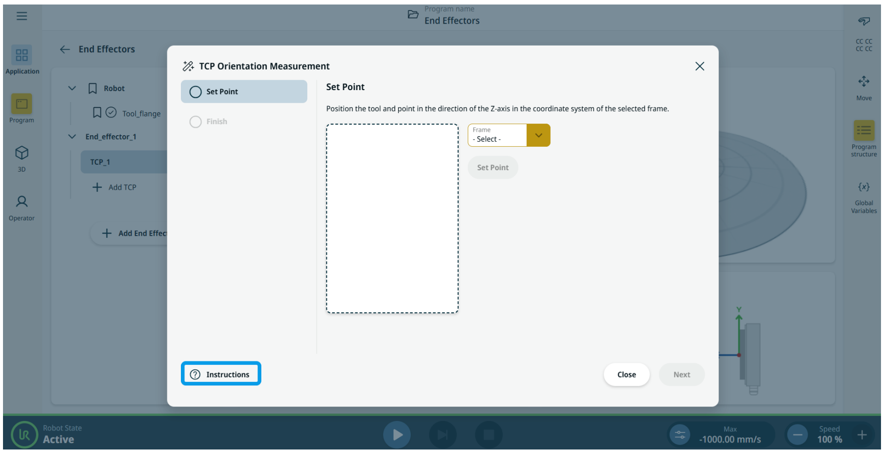
The new TCP orientation wizard provides a convenient way of determining the orientation (rotation) of a TCP. By having the user move the robot to a position where the tool's Z-axis aligns with a selected frame, the wizard can compute the TCP orientation automatically.

TCP Orientation wizard

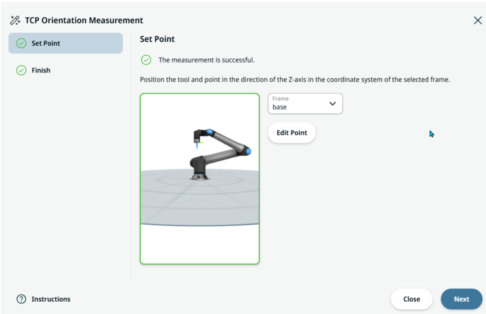
1. On the central panel, tap the TCP Orientation wizard icon,

   Orientation  and the pop-up

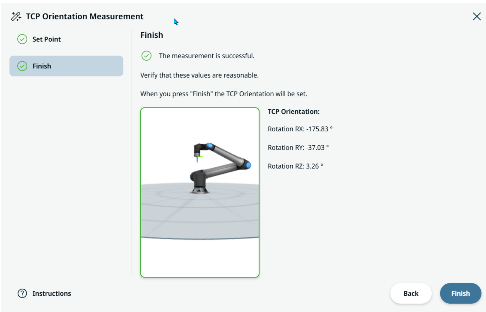
   TCP Orientation Measurement screen appears.



2. Tap **? Instructions** on the bottom left part of the screen.
3. Position the Z-axis of the tool parallel with the Z-axis of the selected frame, for example the base.
4. Tap **Set Point** and jog the robot per prior instruction, then **Save**.
5. When the robot is properly jogged in 3D view, the set point is successful, and tap **Next**.



6. To finish, the TCP orientation measurement, verify the RX, RY, and RZ values.



20.4.3. Payload and Center of Gravity

Description

The new Payload Estimation wizard allows the robot to help the user set the correct payload and center of gravity (CoG) of a work piece, such as a tool or a combination of attachments to the tool flange.

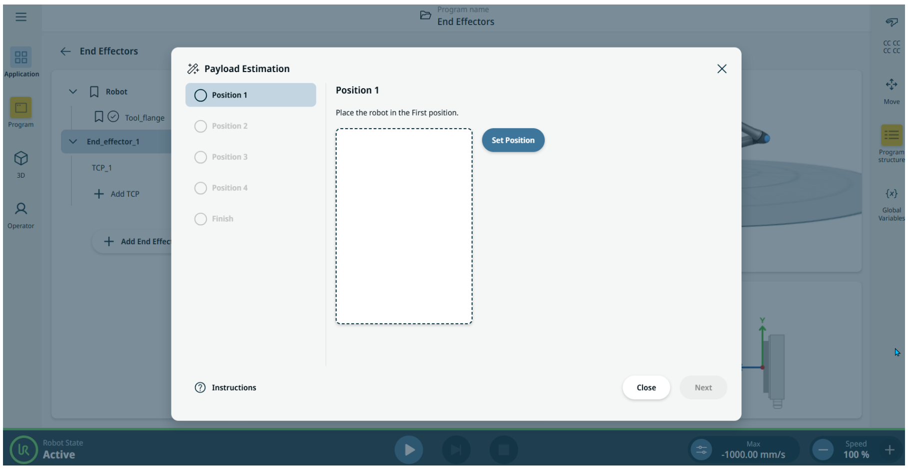
Payload and Center of Gravity (CoG) wizard

1. On the left panel, tap End_effector_1.
2. On the central panel, tap the Payload wizard icon,

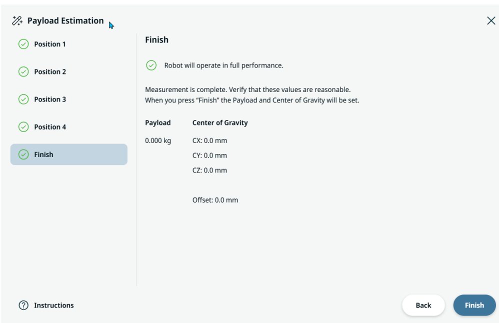
   Payload

   , and the pop-up

   Payload Estimation screen appears.

3. By having the user teach four different positions, the robot can estimate the payload mass and CoG automatically. The wizard's final step will indicate, if the computed payload mass or the combination of payload mass and CoG offset may cause reduced robot performance.

As in the TCP position wizard, the user will be informed, if positions need to be re-taught, because the taught positions are not sufficiently diverse.
A pop-up with instructions can be opened at any time as in the TCP wizards.

NOTICE

Follow the these guidelines for best Payload Estimation results:

- Ensure the TCP positions are as different as possible from each other
- Perform the measurements within a short time span
- Avoid pulling on the tool and/or attached payload before and during estimation
- Robot mounting and angle must be correctly defined in the installation

Renaming, Duplicating, and Deleting Payloads

You can start configuring a new Payload with the following actions:

- Tap to rename a Payload.
- Tap to duplicate a Payload.
- Tap to delete a selected Payload. You cannot remove the last Payload.

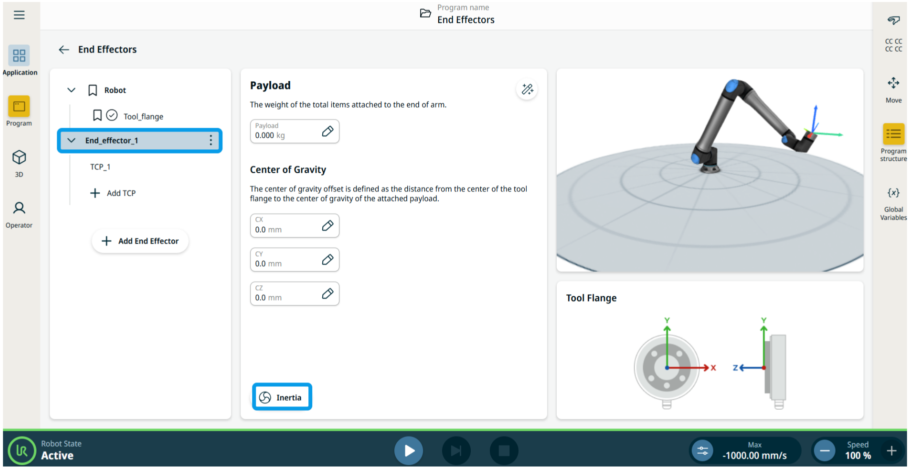
20.4.4. Payload Inertia

Description

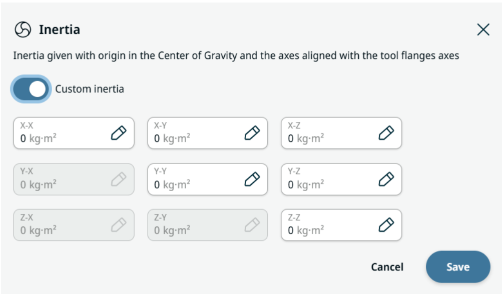
Configuring the inertia of heavy payloads enhance the robot's motion performance and reduces risk of vibrations and protective stops.

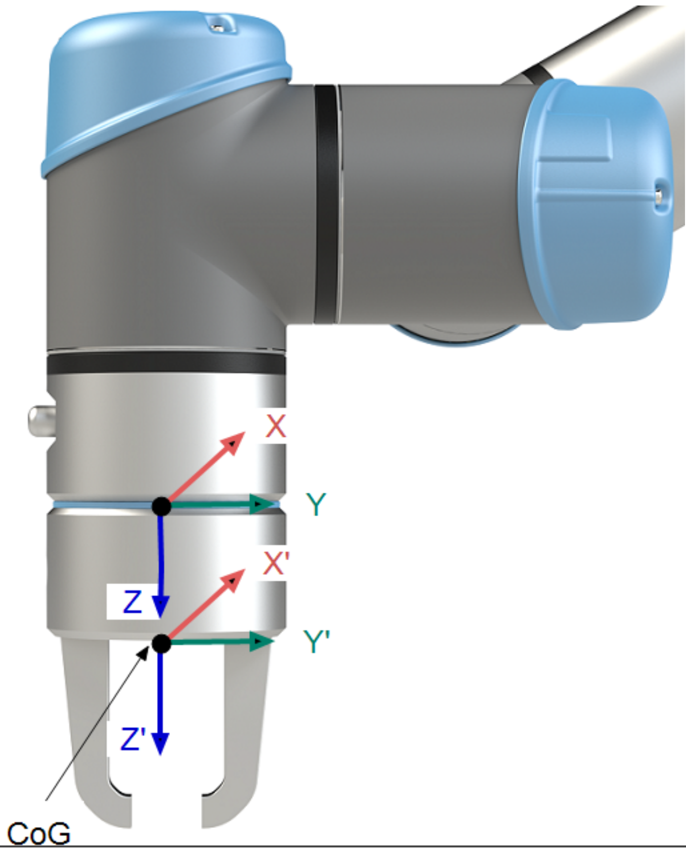
Setting Inertia Values

1. Tap **End_effector_1** in the left panel.
2. In the central panel, tap the **Inertia icon** below the Center of Gravity fields.



3. You can select **Custom Inertia** to set inertia values. Slide the button to activate.
4. Tap the fields: **X-X**, **X-Y**, **X-Z**, **Y-Y**, **Y-Z**, and **Z-Z** to set the inertia for the selected payload. **Save**.





20.5. Motion Profiles

Description

Motion Profiles application allows you to share move parameters across your program by defining speed and acceleration with a profile name. The profile name can then be referred in any new move node in your program, even if the moves are not grouped in a folder. This application applies to the following commands:

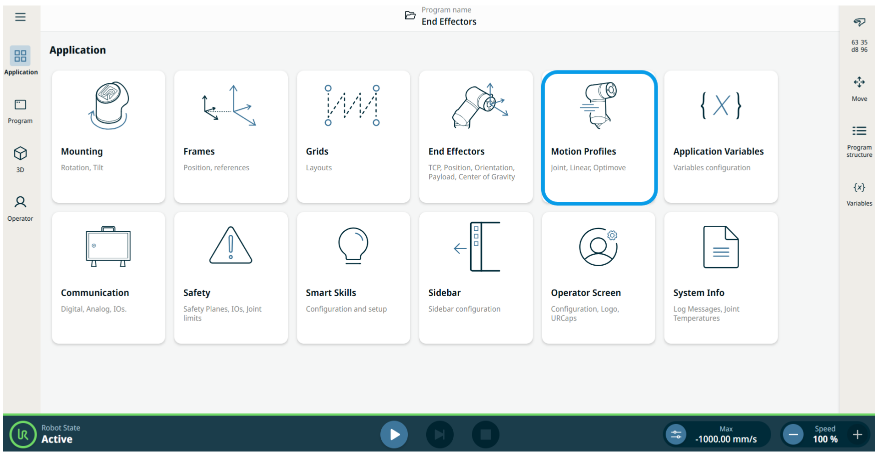
- Joint Move
- Linear/Direction Move
- Process/Circular Move

Intended programming flow

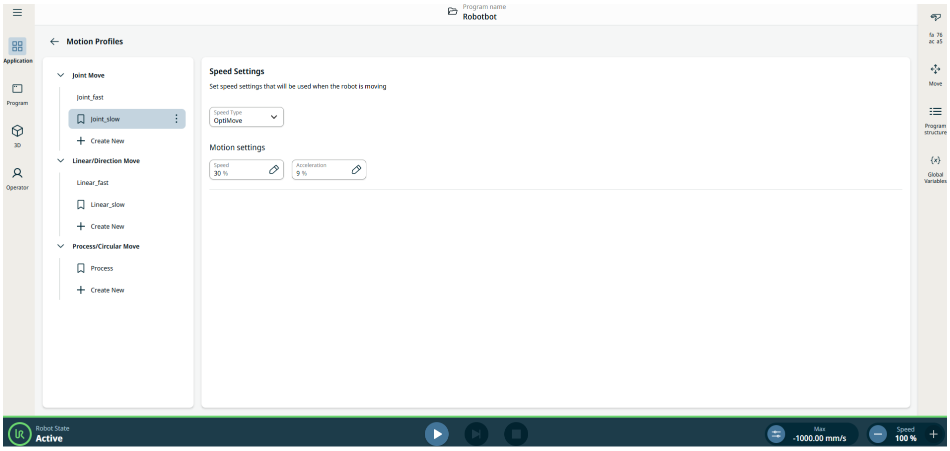
1. User starts by setting up all the Motion Profile needed for the application.
2. Choose a suitable name for easy recognition.
3. Choose the proper Move type (Joint, Linear/Direction, Process Move, Circular arc) from the commands toolbox when the Move node is inserted .
4. Every Move node should use a Motion Profile for easy overview.

Using the Motion Profiles application functionality

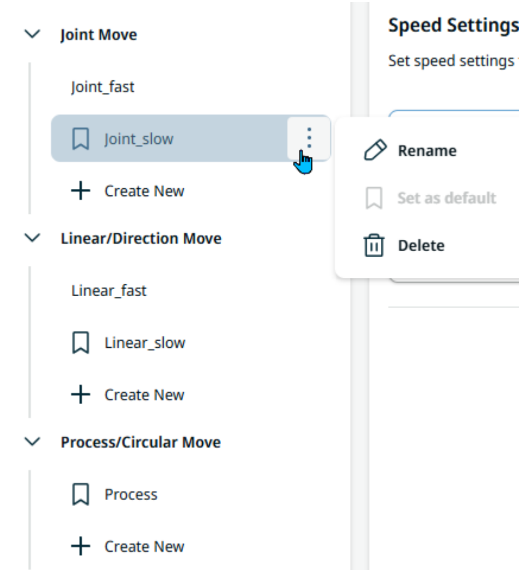
1. Go to the application nodes screen. See Application Tab.
2. Tap the **Motion Profiles** icon.



The Motion Profiles screen appears, which is divided into two panels: left panel contains the three move types, and right panel contains the speed settings.



Left Panel



The left panel contains three move types, which correspond to the three motion profile categories:

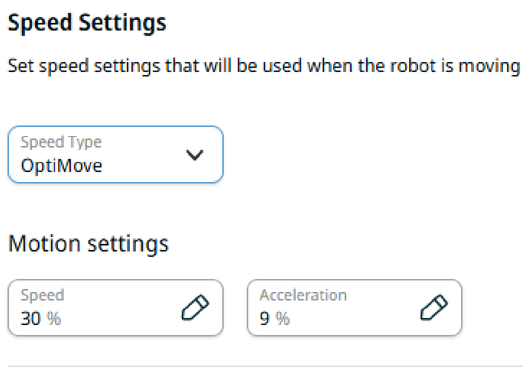
- Joint Move
- Linear/Direction Move
- Process/Circular Move

Note: Derived program nodes will use the profile for their underlying move type.
16.3.15 Joint Move on page 156 for joint profile.
16.3.6 Direction on page 137 node for linear profile.
16.3.4 Circular Arc Move on page 126 for process profile.

1. Tap the **kebab icon** and choose **Rename**, **Set as Default**, or **Delete**.
2. Tap the appropriate **+ Create New** action in each profile to add a new motion profile.

   Note: The last profile of any move type cannot be deleted.

Right Panel



The right panel contains three fields of the speed settings:

- Speed Type
- Speed
- Acceleration

1. Tap the **Speed Type** field and choose either **OptiMove** or **Classic**.

   OptiMove. This is the recommended option for program safety and reliability, which allows the values to be set as a percentage of the allowed maximum. See OptiMove.

   Classic. This speed type enables you to specify values in m/s and m/s².
2. Tap the **Speed** field, and three tabs appear:
   - Value
   - Variable
   - Expression
3. Enter the data in each tab and tap **Confirm**. See Value-Variable-Expression Tabs.
4. Tap the Acceleration field, and three tabs appear:
   - Value
   - Variable
   - Expression
5. Enter the data in each tab and tap **Confirm**. See Value-Variable-Expression Tabs.

Example Program for Motion Profiles

1. Go to steps 1-2 of "Using the Motion Profiles application functionality."
2. On the left panel, tap Process/Circular Move.
3. Tap + Create New, type the motion profile name as "Welding_speed." Save.
4. On the right panel, change the motion settings speed to 50 in the Value tab, and Confirm.
5. Tap the kebab icon of "Welding_speed" and set it as default.
6. Tap the kebab icon of Process, rename it "Medium_speed" and Save.
7. On the right panel, change its motion settings speed to 100 in the Value tab, and Confirm.
8. Tap again + Create New, type the motion profile name "Fast_speed" and Save.
9. On the right panel, change the motion settings speed to 150 in the Value tab, and Confirm.
10. Go to the main program screen. See Command Nodes.
11. Tap the Process Move icon and choose Welding_speed as Point_1.
12. Go to Edit Point button and jog the robot using the X, Y, Z plus-minus button in the multitask screen.
13. Tap the Blend field, enter 5 in the Value tab, and Confirm.
14. Repeat step 10 and tap again the Process Move icon. Choose Medium_speed as Point_2.
15. Repeat step 12.
16. Tap the Blend field, enter 10 in the Value tab, and Confirm.
17. Repeat step 10 and tap again the Process Move icon. Choose Fast_speed as Point_3.
18. Tap the Blend field, enter 15 in the Value tab, and Confirm.
19. In the footer, play the program.

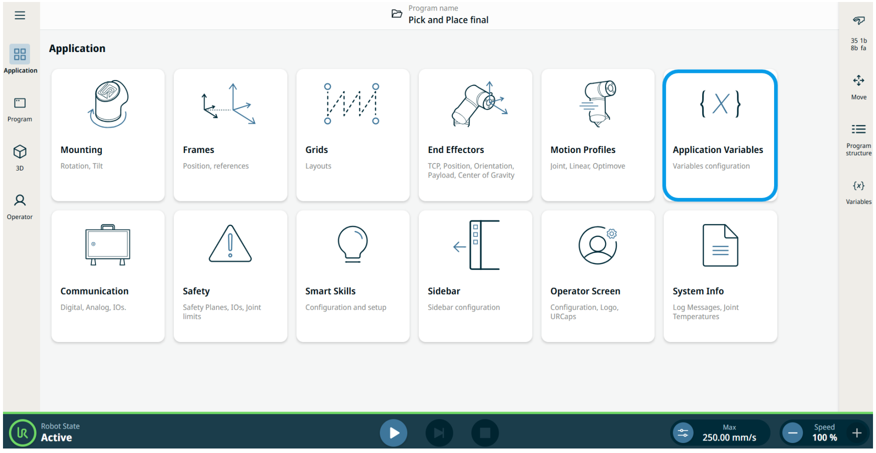
20.6. Application Variables

Description

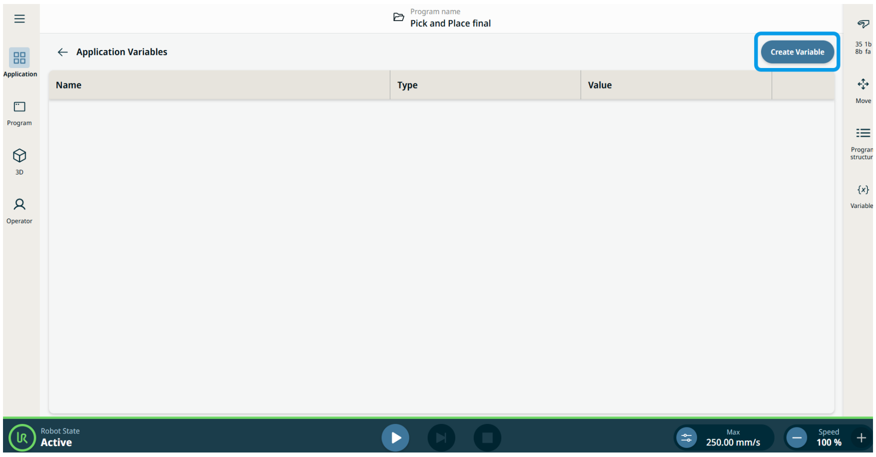
Application Variables allows you to define a list of variables that are persisting between program runs and robot restarts. It can be used in the same way as any other variable(s) in the program. Along with values, variables can be exported together with program.

Using the Application Variables functionality

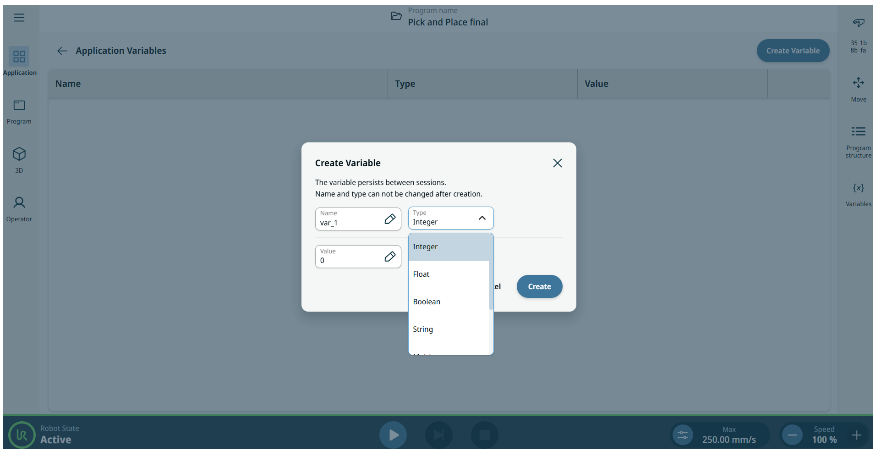
1. Go to the application nodes screen. See Application Tab.
2. Tap the **Application Variables** icon.



3. The Application Variables configuration page displays all created variables for the currently loaded program. Information is sorted in column by name, type, and value.



4. On the screen’s top right side, tap the **Create Variable** button to add a new variable. A pop-up window appears with the name, type, and value fields for configuration.



5. In the **Name** field, enter your variable name. Edit the value in the **Value** field. In the **Type** field, you see the allowed variable types and choose the one pertinent to your program.

20.6.1. Variable Types

Variable Types and examples

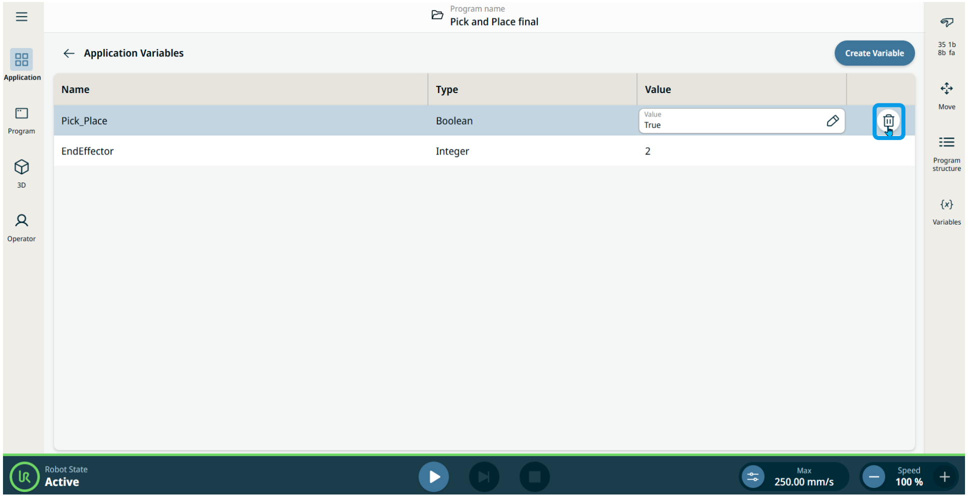
These are the different variable types and their corresponding examples:

- **Integer**: numbers [1, 3.14159]
- **Float**: numbers [3, 4.24261]
- **Boolean**: True, False
- **String**: “Next part”
- **Matrix**: [[1, 0, 0], [0, 1, 0], [0, 0, 1]]
- **Pose**: p[0.1, 0, 0.3, 3.14, 0, 0]
- **Array**: array of numbers, strings, poses - [3, 44, 55, 66, 77], ["Start position"], [p[0, 0, 0, 3.14, 0, 0], p[0.3, -0.1, 0.5, 0, 1.57, -0.33]]

20.6.2. Delete Variables

Deleting variables in programs

Tap the trash button to remove a variable. Removing a variable is possible only when a program is stopped. Program will be stopped automatically if it's still running.



> **NOTICE**
>
> Type and name cannot be changed after variable is added. Value can be changed when application is stopped. Value is also updated live when program is running.

20.6.3. Use Variables in Programs

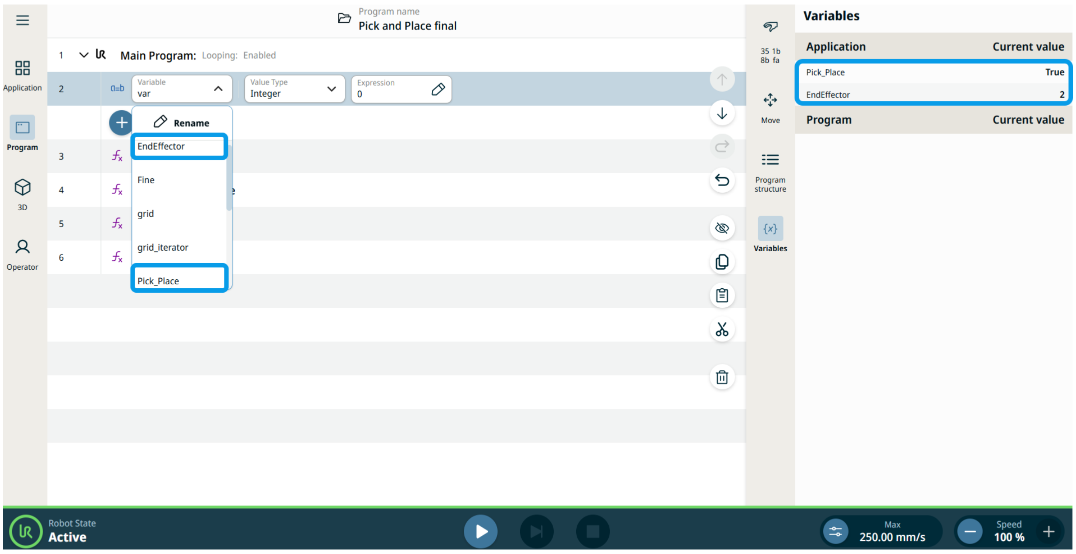
Using variables in programs

> **NOTICE**
>
> Structs, which are complex data types, are not supported in application variables. Communication handler objects (e.g. xmlrpc, ros_subscriber, etc.) are not supported as application variables.

Application variables can be used in the same way as any other variables in the program. They are added to a list of variables available in the dropdown lists.

1. Open the Assignment command node. See Assignment.
2. In the **Variable field**, you can see the variables created in the Application Variables.
3. Tap **Variables** in the sidebar to view the created variables.

> **NOTICE**
>
> See the variables inside the blue box in both the main screen and the multitask screen. Information found in both screens correspond to each other.

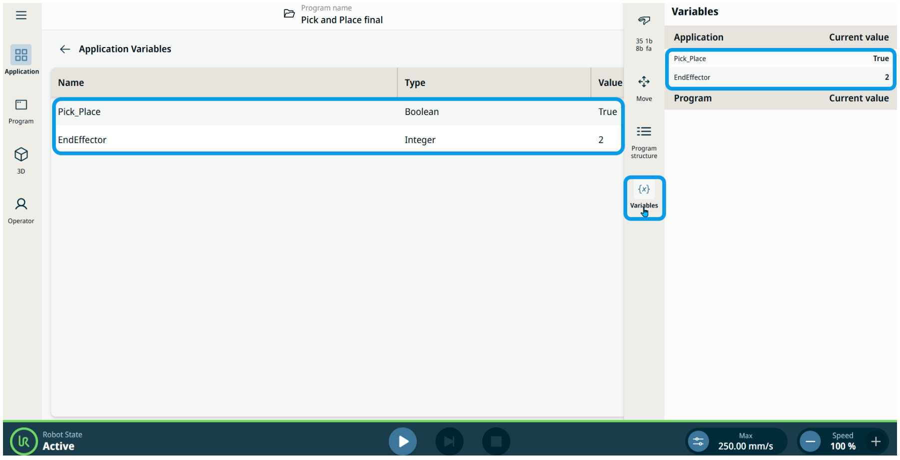


20.6.4. Showing Variables

Showing variable values

Current variable value is shown both on the sidebar in variables section and on the Application Variables screen in Application section.

1. Go to the Application Variables configuration screen, and you see the created variables.
2. Tap **Variables** in the sidebar. The multitask screen expands to the center, and you see the created variables with their currents value in the Application list.



20.6.5. Saving Variables

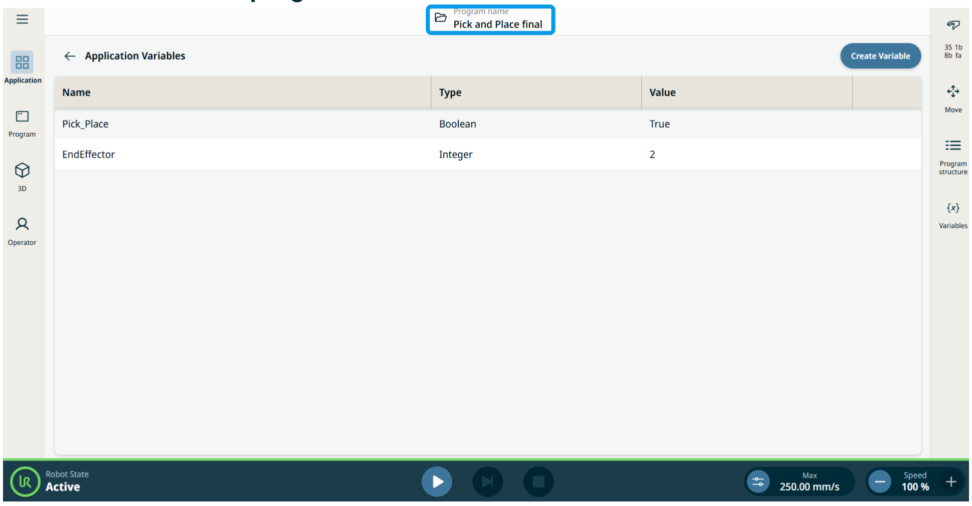
Saving variables

Each program in PolyScope X has its own set of application variables. Variables are saved to the internal program database when program is stopped and automatically every ten minutes.

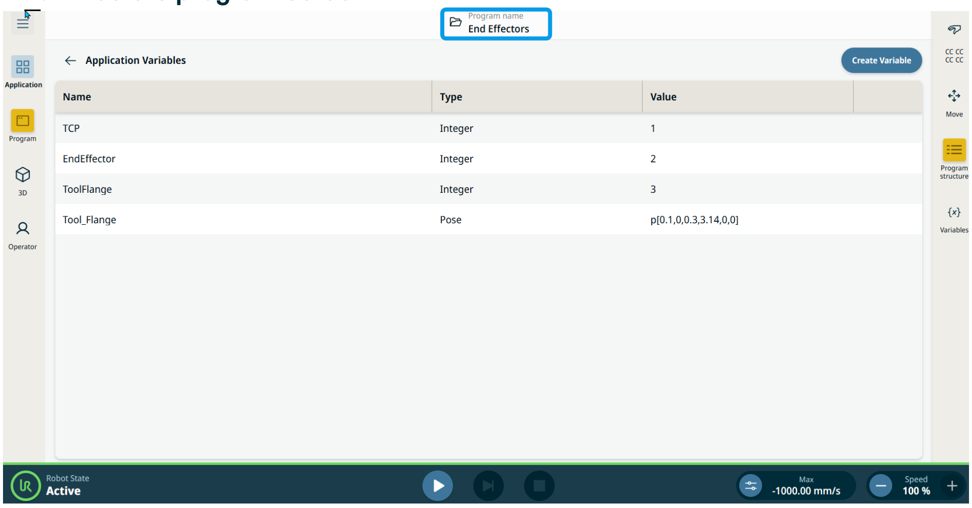
> **NOTICE**
>
> Every time a program is switched, all variables are cleared and replaced with variables defined in the newly loaded program. When a program is loaded again, values of Application Variables are automatically restored to the last state.

See the switch of the application variables between the programs “Pick and Place final” and “End Effectors.”

Pick and Place final program screen



End Effectors program screen



20.7. Communication

Description

The Communication application allows you to monitor and set the live IO (input-output) signals from/to the robot control box.

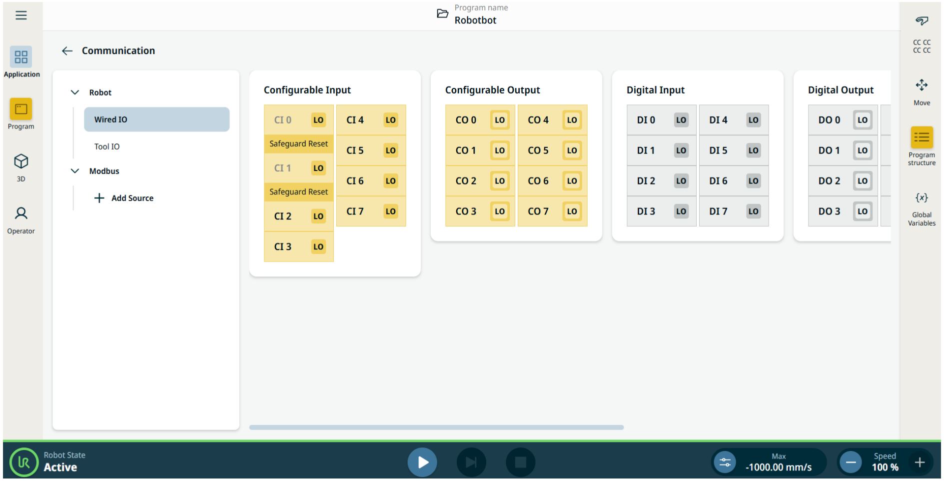
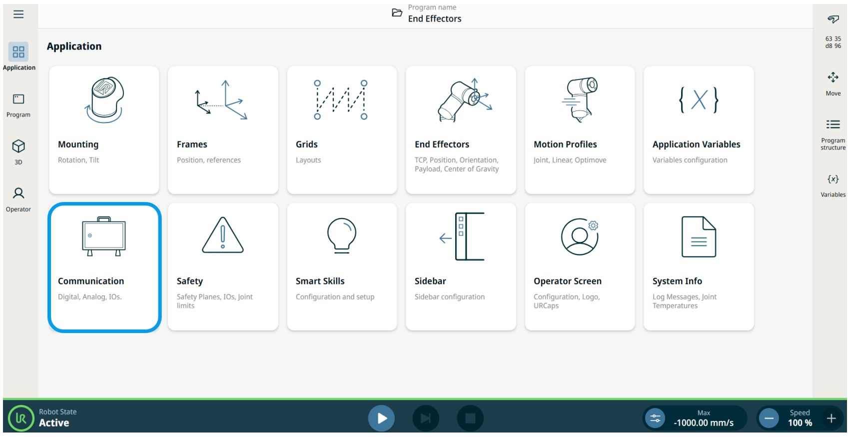


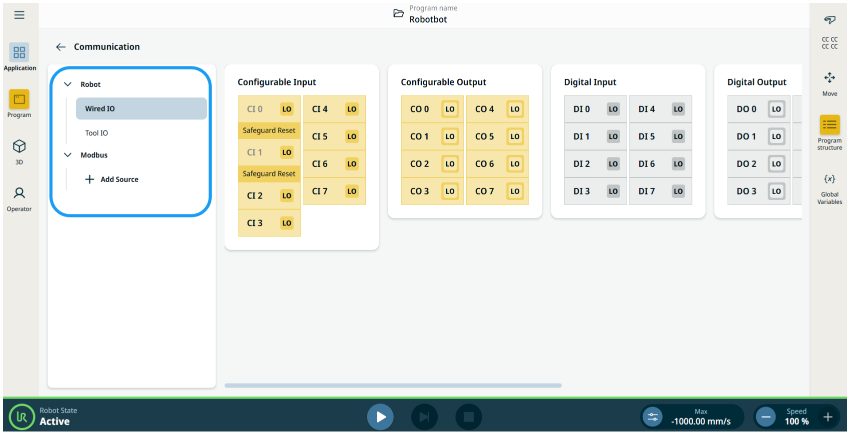
Figure 1.3: *Communication screen displaying IOs.*

Using the Communication application functionality

1. Go to the Application menu screen. See Application Tab.
2. Tap the **Communication** icon.



The Communication screen appears, which is divided into the left panel and the right panels.



The left panel consists of the communication tree, composed of three functions:

- **Robot Wired IO**
- **Robot Tool IO**
- **Modbus**

The right panels show the editable fields of the chosen function.

20.7.1. Wired IO

Description

Wired IO is used to monitor and set the live IO signals from/to the control box.

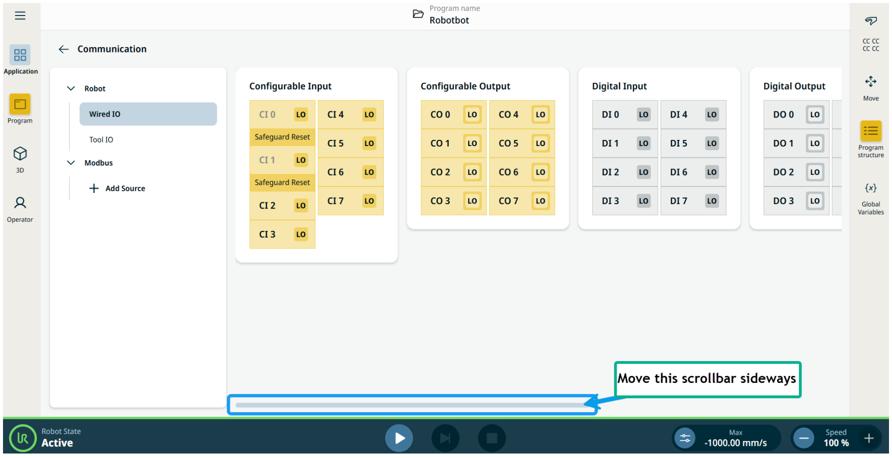
To access Wired IO

1. Tap **Wired IO** on the left panel.

   The Communication screen displays the current state of the IO, including during program execution. If anything is changed during program execution, the program stops. At program stop, all output signals retain their states.

2. Use the scrollbar on the right panel to move sideways, and you can see the editable fields of the six components of the wired IO:

   - Configurable Input
   - Configurable Output
   - Digital Input
   - Digital Output
   - Analog Input
   - Analog Output



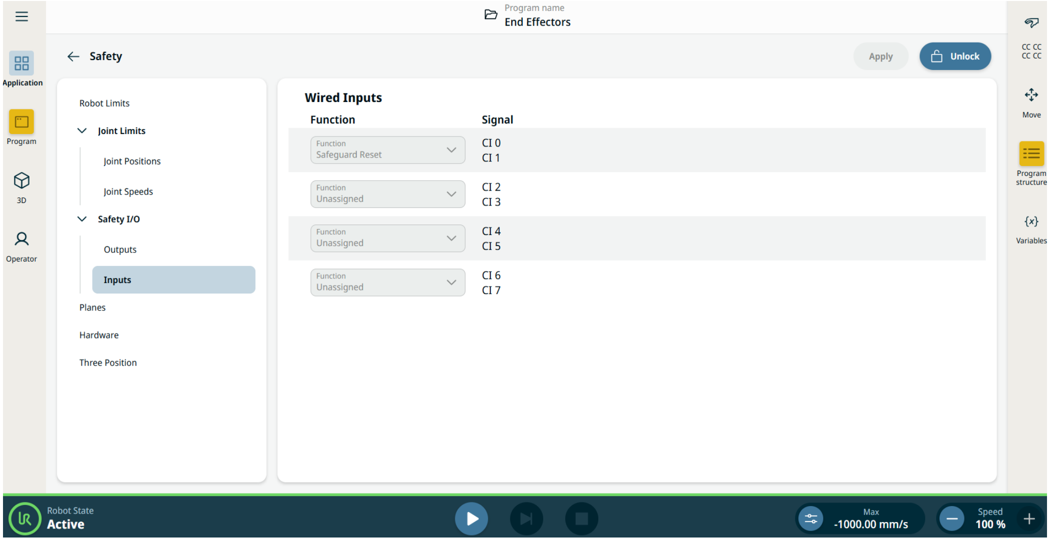
Input is used to start, stop, or pause the program, or as an external **Freedrive** button.

Output shows whether the robot is running or not.

I/O Setup

Description

Use the I/O Setup screen to define I/O signals and configure actions with the I/O tab control. The types of I/O signals are listed under **Input** and **Output**.



> **NOTICE**
> When starting programs from an I/O or fieldbus input, the robot can begin movement from the position it has, there will not be any manual movement to the first waypoint via PolyScope required.

Inputs

1. To the Application .
2. Go to Safety.
3. Tab the Inputs in the Safety I/O section.
4. Unlock the settings.
5. Assign a function to the signal group.

See the Safety I/O Signals for a description of the safety function.

Available Input Actions

Command	Action
Start	Starts or resumes the current program on a rising edge (only enabled in Remote Control)
Stop	Stops the current program on a rising edge
Pause	Pauses the current program on a rising edge
Freedrive	When the input is high, the robot goes into Freedrive (similar to the **Freedrive** button). The input is ignored if other conditions disallow Freedrive.

WARNING

If the robot is stopped while using the Start input action, the robot slowly moves to the first waypoint of the program before executing that program. If the robot is paused while using the Start input action, the robot slowly moves to the position from where it was paused before resuming that program.

Outputs

1. To the Application .
2. Go to Safety.
3. Tab the Outputs in the Safety I/O section.
4. Unlock the settings.
5. Assign a function to the signal group.

You can enable OSSD on each output signal.
See the Safety I/O Signals for a description of the safety function.

Available Output Actions

Action	Output state	Program state
Low when not running	Low	Stopped or paused
High when not running	High	Stopped or paused
High when running, low when stopped	Low High	Running, Stopped or paused
Low on unscheduled stop	Low	Program terminated unscheduled
Low on unscheduled stop, otherwise High	Low High	Program terminated unscheduled Running, stopped or paused
Continuous Pulse	Alternates between high and low	Running (pause or stop the program to maintain the pulse state)

Program Termination Cause

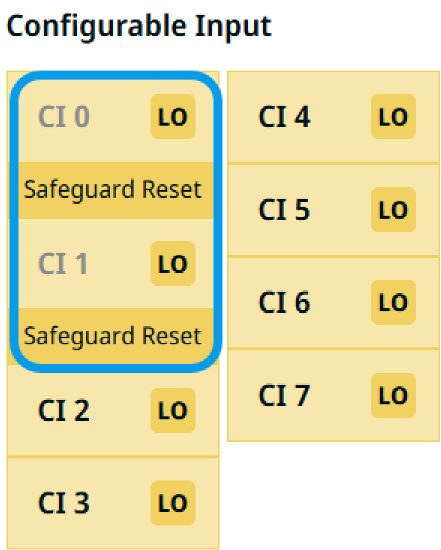
An unscheduled program termination can occur for any of the reasons listed below:

- Robot stop
- Fault
- Violation
- Runtime exception

Configurable Input

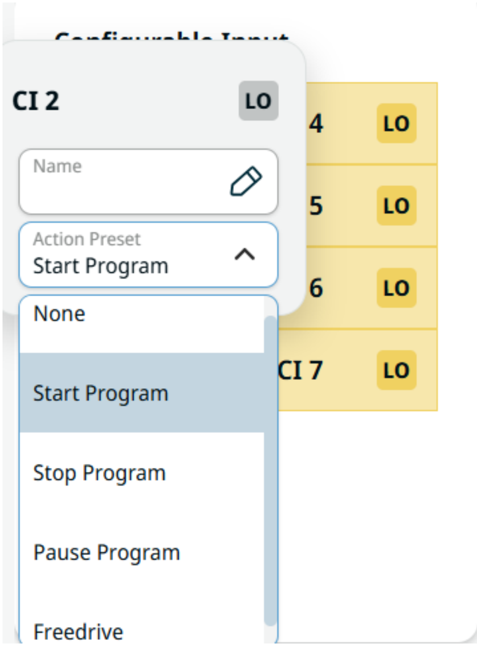
Description

Configurable input can be reserved for special safety settings defined in the IO Setup under Safety IO. Under those which are reserved for safety settings (CI 0 and CI 1), named **Safeguard Reset**, are in gray and cannot be edited. CI 2-CI 7 have editable fields.



To access Configurable Input

1. Tap **CI 2**, and the **Name** and **Action Preset** fields appear. In the **Name** field, create a name and **Confirm**. The new name can now be seen in the communication screen.



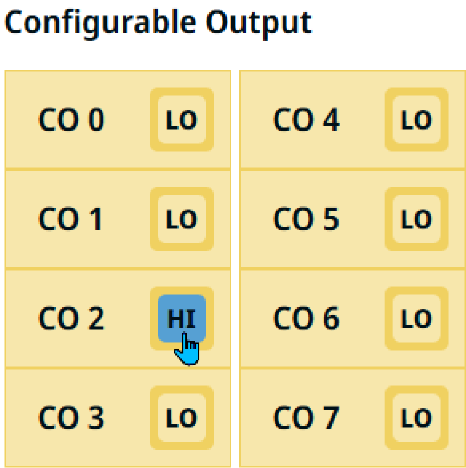
2. Tap the **Action Preset** field, and five choices are available:
   - **None**
   - **Start Program**. Starts the loaded program.
   - **Stop Program**. Stops the loaded program.
   - **Pause Program**. Pauses the loaded program.
   - **Freedrive**. Puts the robot in Freedrive mode, similar to the button on the back of the Teach Pendant.
3. Choose the pertinent action preset.
4. Repeat the steps above for CI 3 to CI 7.

Configurable Output

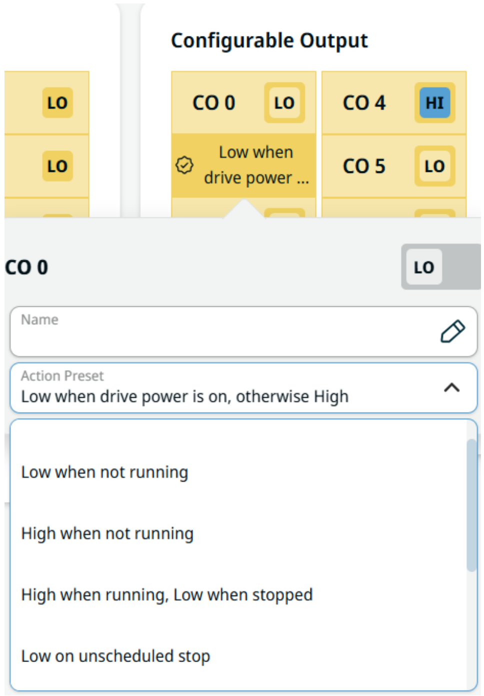
Description

The **Configurable Output** field ranges from CO 0 to CO 7.

To access Configurable Output



1. You can click the square button beside each CO to choose either LO or HI.
2. Tap **CO 0**, and the **Name** and **Action Preset** fields appear.
3. In the **Name** field, create a name and **Confirm**. The new name can now be seen in the communication screen.



4. Tap the **Action Preset** field, and nine choices are available:
   - **None**
   - **Low when not running**. Output state is LO. Program state is stopped or paused.
   - **High when not running**. Output state is HI. Program state is stopped or paused.
   - **High when running, Low when stopped**. Output state is LO then HI. Program state is running, stopped, or paused.
   - **Low on unscheduled stop**. Output state is LO. Program state is terminated unscheduled.
   - **Low on unscheduled stop, otherwise High**. Output state is LO then HI. Program state is terminated unscheduled, running, stopped, or paused.
   - **Continuous Pulse**. Output state alternates from LO to HI. Program state is running.
   - **High when drive power is on, otherwise Low**.Output state is HI.
   - **Low when drive power is on, otherwise High**. Output state is LO.
5. Choose the pertinent action preset.

   By selecting the last two action presets, the HI/LO value of signals will be changed when power is OFF/ON.

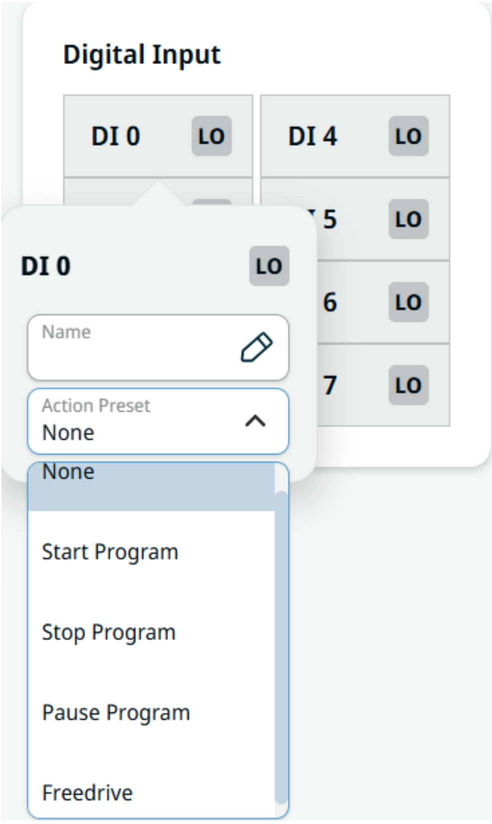
Digital Input

Description

The **Digital Input** field ranges from DI 0 to DI 7.

To access Digital Input

1. Tap **DI 0**, and the **Name** and **Action Preset** fields appear.
2. In the **Name** field, create a name and **Confirm**. The new name can now be seen in the communication screen.



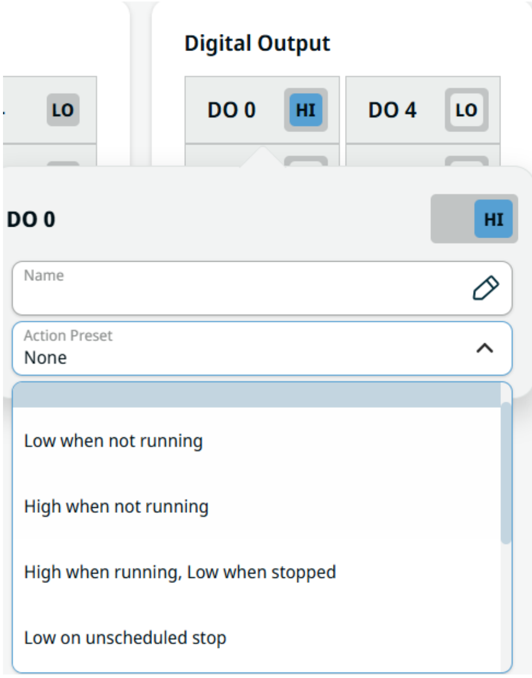
3. Repeat steps 3 onward of Configurable Input for DI 1 to DI 7.

Digital Output

Description

The **Digital Output** field ranges from DO 0 to DO 7 and are set independently to either high or low.

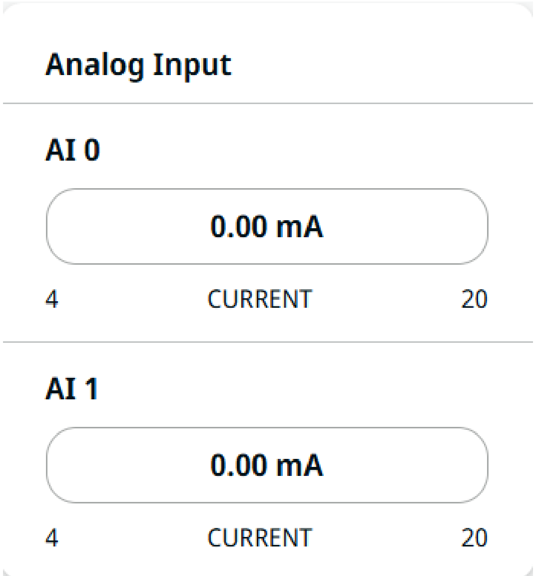
To access Digital Output Follow all the steps as in Configurable Output.



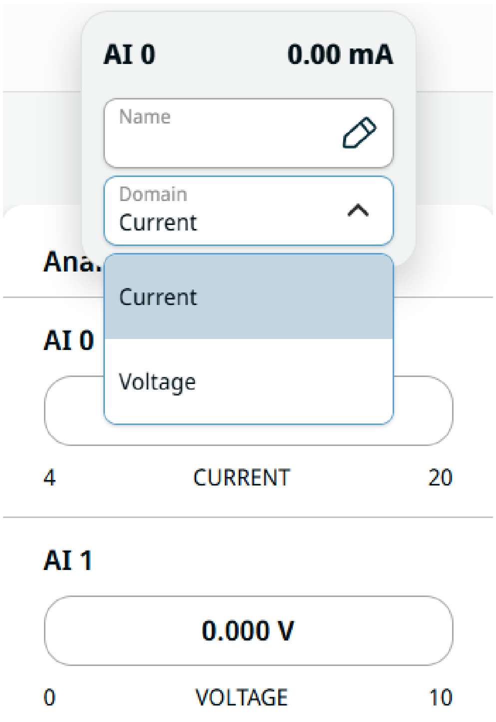
Analog Input

Description Analog input has two fields for AI 0 and AI 1. It can be set to 4-20 mA. These settings are persistent over restarts of the robot controller and saved in the installation.

To access Analog Input



1. Tap **AI 0**. The **Name** and **Domain** fields appear.
2. In the **Name** field, create the name you prefer.



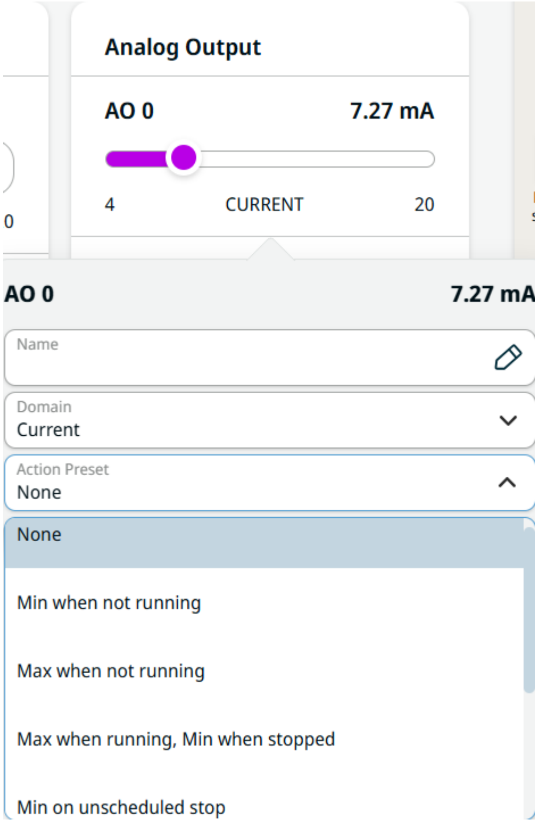
3. Tap the **Domain** field, and you can choose either Current or Voltage.
4. Tap **AI 1**.
5. Repeat steps 2-3.

Analog Output

Description

Analog output has two fields for AO 0 and AO 1. As with analog input, it can be set to 4-20 mA.

To access Analog Output



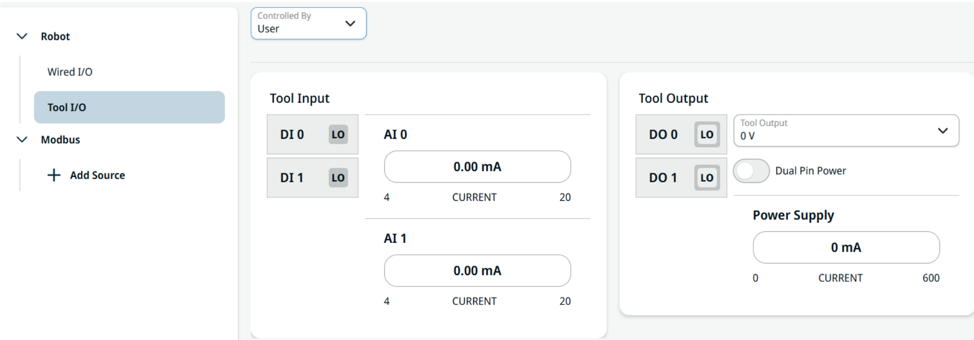
1. Tap the purple slider of each field to adjust the current or voltage settings.
2. Tap **AO 0**.
3. In the **Domain** field, you can choose either Current or Voltage.
4. Tap the **Action Preset** field, and eight choices are available:
    - None
    - Min when not running
    - Max when not running
    - Max when running, Min when stopped
    - Min on unscheduled stop
    - Min on unscheduled stop, otherwise Max
    - Max when drive power is on, otherwise Min
    - Min when drive power is on, otherwise Max
5. Choose the pertinent action preset. The purple slider reflects the action preset you choose.
6. Tap **AO 1** and repeat steps 3-5.

   By selecting the last two action preset, the Min/Max value of signals will be changed when power is OFF/ON.

20.7.2. Tool I/O

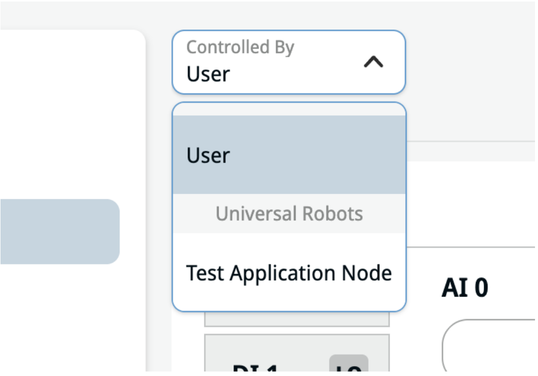
Description Tool I/O monitors and sets the live IO signals from/to the control box.

To access Tool IO Tap **Tool I/O** on the left panel. You can see the editable fields of Tool Input and Tool Output on the right.



URCaps in Control dropdown

The **Controlled By** dropdown field allows user to select a URCap that can control the tool settings. If a URCap is installed that has this capability, it will be displayed in the list. Follow here on how to install a URCap.



When selected, the URCap will be responsible for the following settings:

- **Tool Output**

  Setting can be 0, 12, or 24 V.
- **Dual Pin Power**

  Setting is either true or false.
- **Domain for analog inputs**

  Setting current/voltage per analog input, as seen in the pop-up when pressing one.
- **Power Output for digital outputs**

  Setting Sinking/Sourcing/Push/Pull, as seen in the pop-up when pressing one.
- **Tool Communication settings**

  These settings are not available on the user interface, but are the enabled, baud rate, parity, stop bits, rx idle chars, tx idle chars.

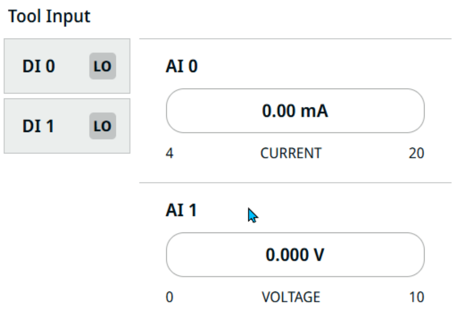
Upon selection of a URCap, any controls related to the above settings will be disabled. The new settings will be applied when an entry is selected in the dropdown, when the program is loaded, and when the program is played.

Tool Input

Description

The **Tool Input** is composed of four fields:

- DI 0
- DI 1
- AI 0
- AI 1



To access Tool Input

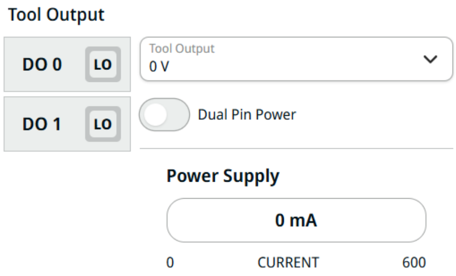
1. Tap **DI 0**, and the **Name** and **Action Preset** fields appear.
2. In the **Name** field, create a name and **Confirm**.
3. Tap the **Action Preset** field and select from the five available choices.
4. Repeat steps 1-3 for **DI 1**.
5. Tap **AI 0**. A **Name** and **Domain** fields appear.
6. In the **Name** field, create the name you prefer.
7. Tap the **Domain** field, and you can choose either Current or Voltage.
8. Repeat steps 5-7 for **AI 1**.

Tool Output

Description

The **Tool Output** is composed of five fields:

- DO 0
- DO 1
- Tool Output
- Dual Pin Power
- Power Supply



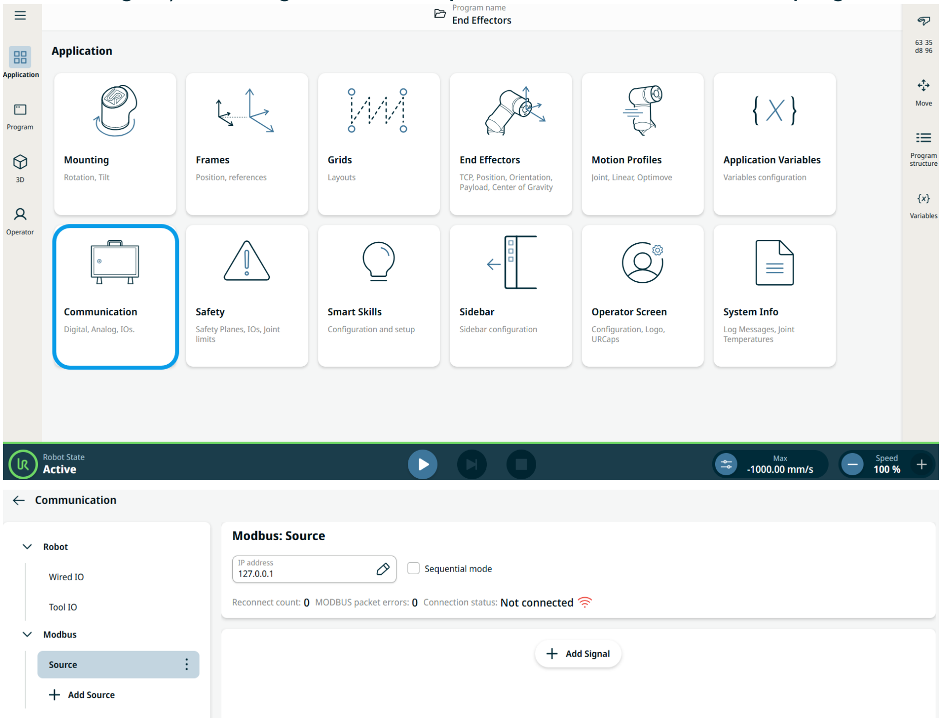
To access Tool Output

1. You can click the **square button** beside each DO to choose either LO or HI.
2. Tap **DO 0**, and three fields appear:
   - Name
   - Action Preset
   - Power Output
3. In the **Name** field, create the name and **Confirm**.
4. Tap the **Action Preset** field and nine choices are available.
5. Choose the pertinent action preset.
6. Tap the **Power Output** field, and four choices are available:
   - **Disabled**
   - **Sinking (NPN)**. When the output is off, the pin allows a current to flow to the ground.
   - **Sourcing (PNP)**. When the output is on, the pin provides a positive voltage source.
   - **Push/Pull**. When the output is on, the pin provides a positive voltage source.
7. Choose the pertinent power output.
8. Repeat the steps for **DO 1**.
9. Tap the **Tool Output** field and choose among the three options available: 0 V, 12 V, 24 V.
10. Dual Pin Power is used as a source of power for the tool. Tap to enable the **Dual Pin Power**. The default tool digital outputs are disabled.

20.7.3. MODBUS

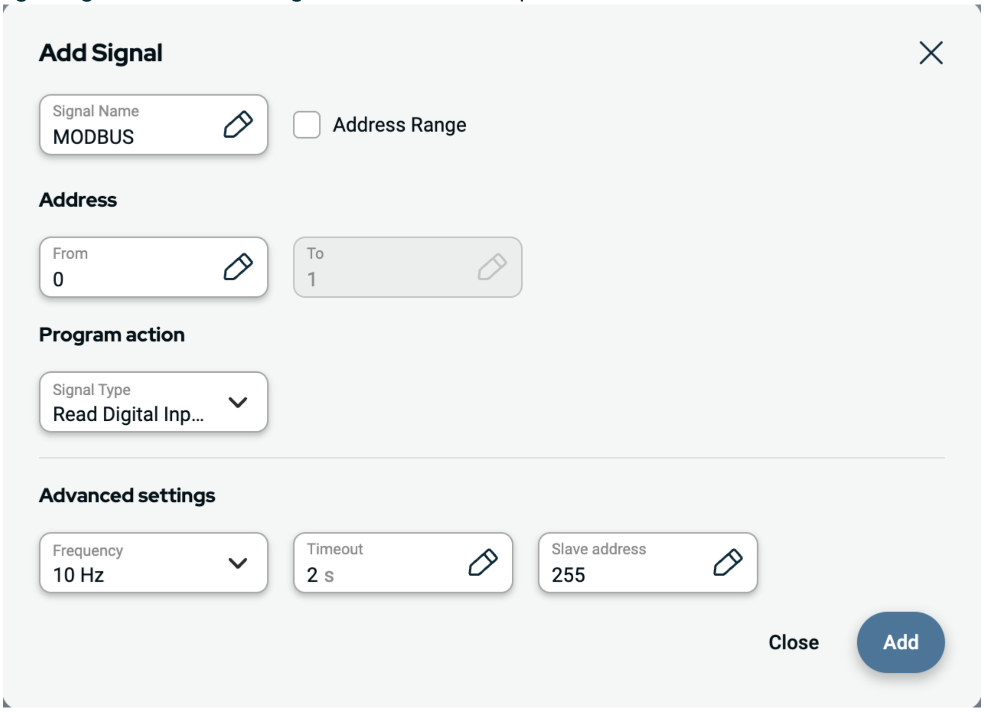
Description

Here, the MODBUS client (master) signals can be set up. Connections to MODBUS servers (or slaves) on specified IP addresses can be created with input/output signals (registers or digital). Each signal has a unique name so it can be used in programs.



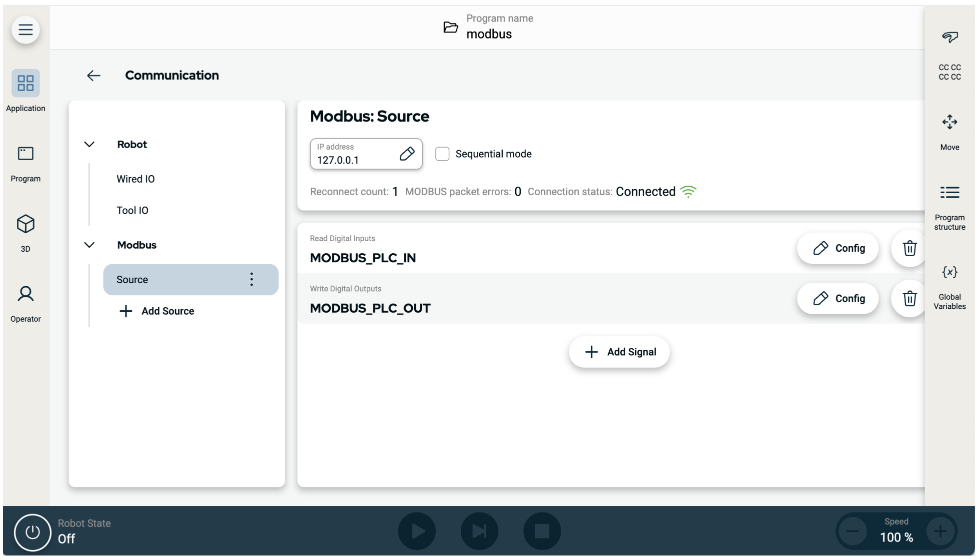
Add Signal

You may customized a signal name when adding a signal.
Select the signal type, signal direction and specify frequency, timeout and other advanced settings. Signal could use single address or multiple addresses.



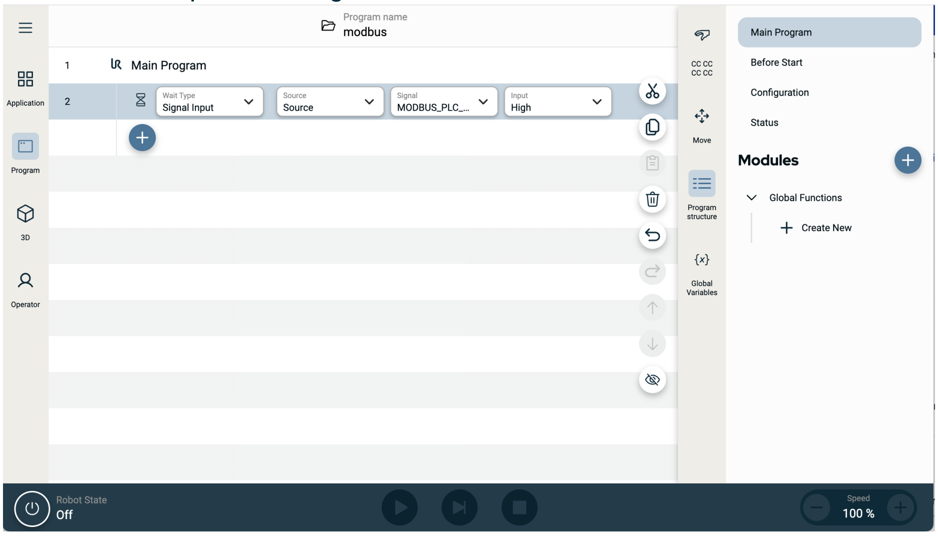
Signal Source

Modbus signal source settings can be edited and deleted, Tap the **Config** buttonto edit, and tap the trashcan icon to delete.

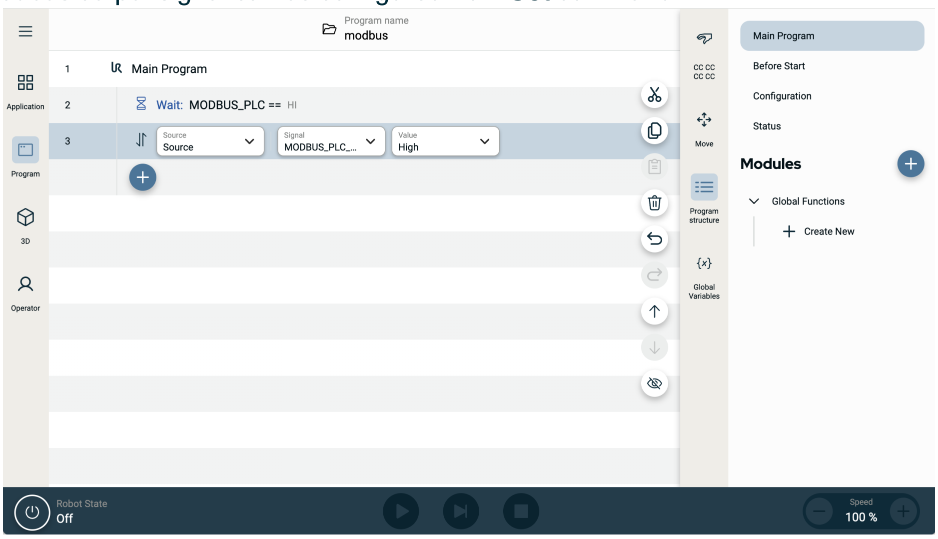


Programming

Similar to other input signals, Modbus signals can be monitored. In the Program, on **Wait** command, select **Signal Input** under **Wait Type**. Then choose the Modbus source, the specific input signal, and the state to wait for.
Address ranges can not be used in logical expressions. The program can use only single address, even if it is part of a range.



A Modbus output signal can be configured from **Set** command.



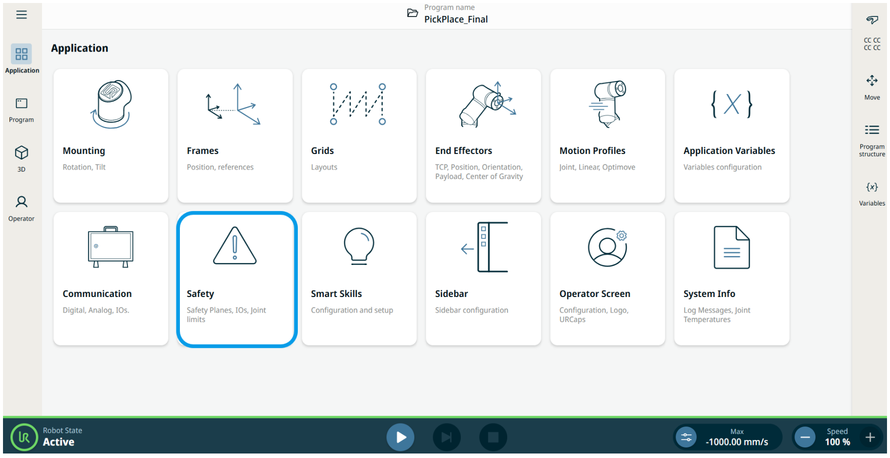
20.8. Safety

Description

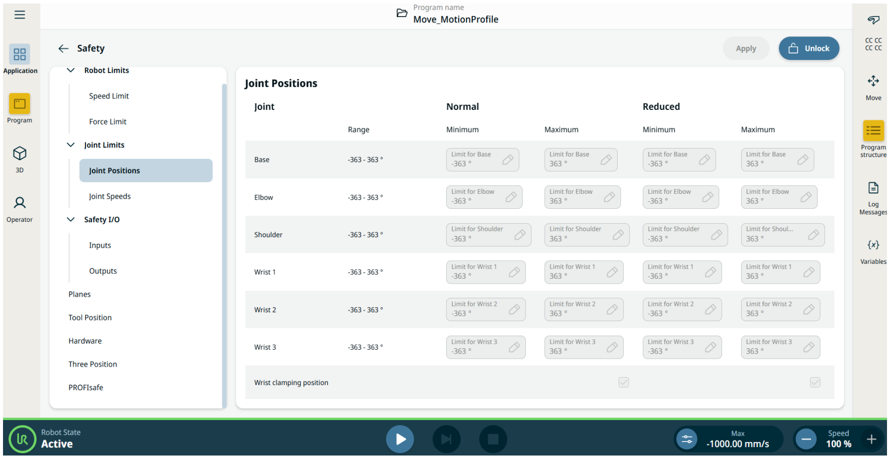
The **Safety** application enables you to access the robot safety settings.

Using the Safety application functionality

1. Go to the Application menu screen. See Application Tab.
2. Tap the **Safety** icon.



The safety screen appears, which is divided into the left panel and the right panel.



The left panel consists of the Safety tree, composed of eight functions:

• Robot Limits	• Safety I/O	• Tool Position	• Three Position
• Joint Limits	• Planes	• Hardware	• PROFIsafe

The right panel shows the editable fields of the chosen function, which are inaccessible.

Unlock safety settings

1. To access the safety settings, tap **Unlock** on the top right side of the safety screen.
2. Enter the safety password.
3. Tap **Confirm** to activate all settings.

20.8.1. Safety Menu Settings

Description

The safety system limits are defined in the Safety Configuration. The safety system receives values from the input fields and detects any violation if any the values are exceeded. The robot controller prevents violations by making a robot stop or by reducing the speed.

> **CAUTION**
> Use of Safety Configuration parameters different from those defined by the risk assessment can result in hazards that are not reasonably eliminated or risks that are not sufficiently reduced.

20.8.2. Robot Limits

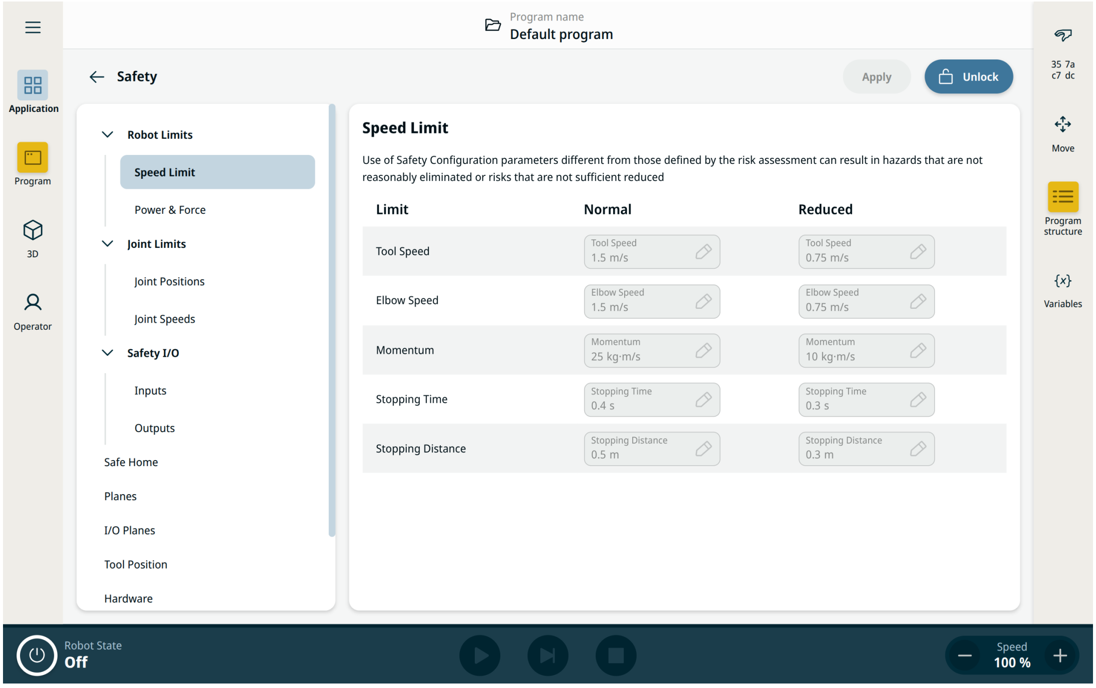
Description

Robot Limits restrict general robot movements.

- Change the values in robot limits (Normal and Reduced) and apply to see the applied changes in safety checksum.
- Verify the values of reduced should be always less than the values in Normal.

> **NOTICE**
> The safety limits restrict forces and motions at the tool flange and the center of the two user-defined tool positions

Speed Limits

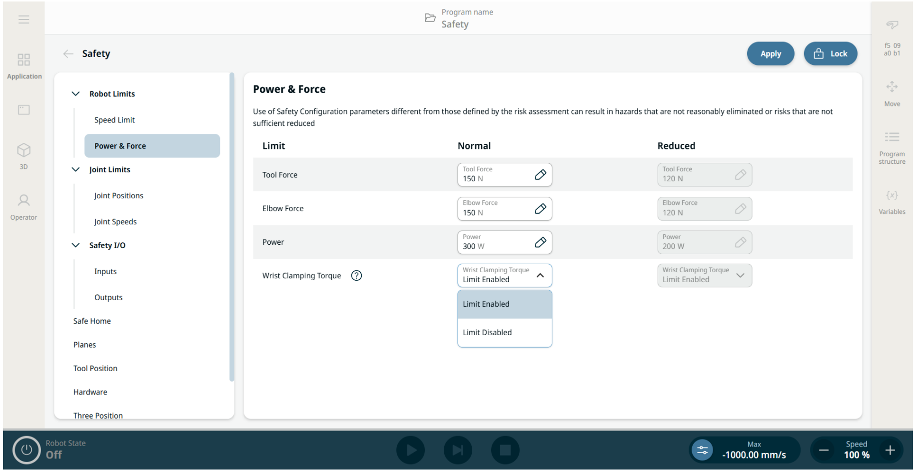


NOTICE

Restricting stopping time and distance affect overall robot speed. For example, if stopping time is set to 300 ms, the maximum robot speed is limited allowing the robot to stop within 300 ms.

Limit	Description
Tool Speed	Limits maximum robot tool speed.
Elbow Speed	Limits maximum robot elbow speed.
Momentum	Limits maximum robot momentum.
Stopping Time	Limits maximum time it takes the robot to stop e.g. when an emergency stop is activated.
Stopping Distance	Limits maximum distance the robot tool or elbow can travel while stopping.

Power and Force



NOTICE

The tool and elbow forces may be exceeded by the three wrist joints if the "wrist clamping torque" safety function is disabled

Limit	Description
Tool Force	Limits the maximum force exerted by the robot tool in clamping situations.
Elbow Force	Limits maximum force that the elbow exerts on the environment.
Power	Limits maximum mechanical work produced by the robot in the environment. This limit considers the payload a part of the robot and not of the environment.
Wrist Clamping Torque	Limits how much torque the wrist can apply for pushing. Enabled by default.

Safety Mode

When no protective stops are active, the safety system operates in a Safety Mode associated with a set of safety limits:

Safety mode	Effect
Normal	This configuration is active by default.
Reduced	This configuration activates when the Tool Center Point (TCP) is positioned beyond a Trigger Reduced mode plane, or when triggered using a configurable input.

Wrist Clamping Torque Limit

Description

Wrist Clamping Torque Limit is a setting that enables or disables limitation of the maximum clamping torque for the wrist joints of the robot. The limitation is enabled by default. This safety function limits how much torque the wrist can apply when pushing.

Wrist Clamping Torque Limit details

When **enabled**, the robot limits torque in the wrist joints to prevent clamping between the wrist, payload, and the lower arm of the robot.

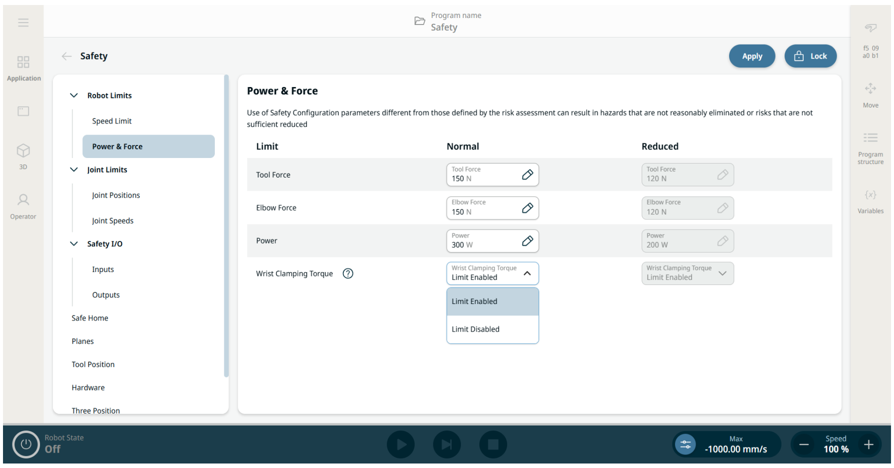
> **NOTICE**
>
> When the setting is enabled, it also defines a non-safety-rated upper limit for the applied torque windows of the base, shoulder, and elbow joints. In force control applications, the achievable contact forces may be significantly lower than the specified forces in the **Tool Force Limit** and **Elbow Force Limit** safety functions.

When **disabled**, the robot can use higher pushing force with the wrist joints, for example in force-controlled applications. When the function is disabled, risks of clamping between the wrist, payload, and lower arm of the robot must be mitigated by other means.

> **NOTICE**
>
> This safety function is enabled on all robots by default. In PolyScope X version 10.12 and newer, it is possible to disable this safety function.

Enable and disable Wrist Clamping Torque Limit

1. Go to the **Application** tab and select **Safety**.
2. Under **Robot Limits**, tap **Power and Force**. Unlock, enter the safety password, and confirm.
3. You can now select **Limit Enabled** or **Limit Disabled** for Normal or Reduced.



4. Tap **Apply** to apply the safety configuration and restart the safety system.
5. A pop-up box appears where you can revert or confirm the configuration.

20.8.3. Joint Limits

Description

Joint limits allow you to restrict individual robot joint movements in joint space by joint rotational position and joint rotational speed. Joint limiting can also be called software-based axis limiting. These are the robot joints:

- Base
- Elbow
- Shoulder
- Wrist 1
- Wrist 2
- Wrist 3

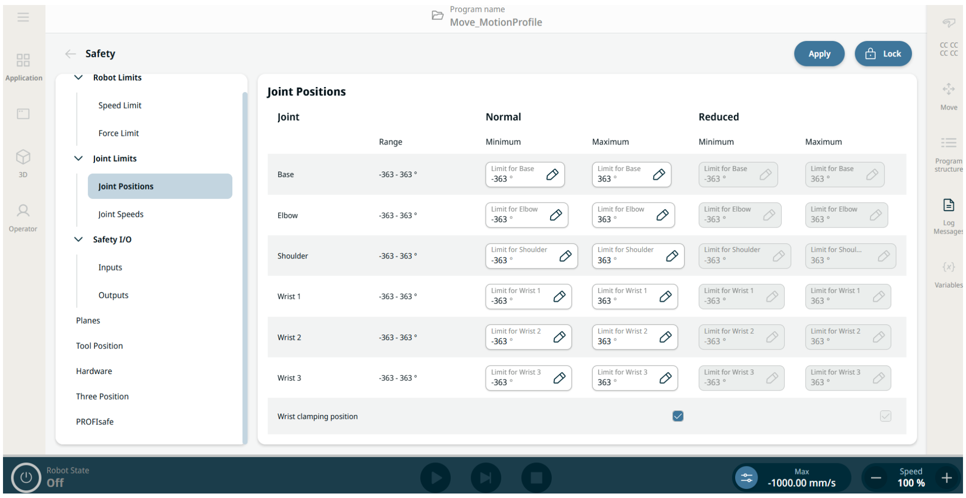
About the Normal and Reduced safety modes, read Robot Limits.

Joint Limits options

There are two Joint Limits options: Maximum speed and Position range.
Maximum speed: defines the maximum angular velocity for each joint.
Position range: defines the position range for each joint. Again, the input fields for Reduced mode are disabled if there is no safety plane or configurable input set to trigger it. This limit enables safety-rated soft axis limiting of the robot.

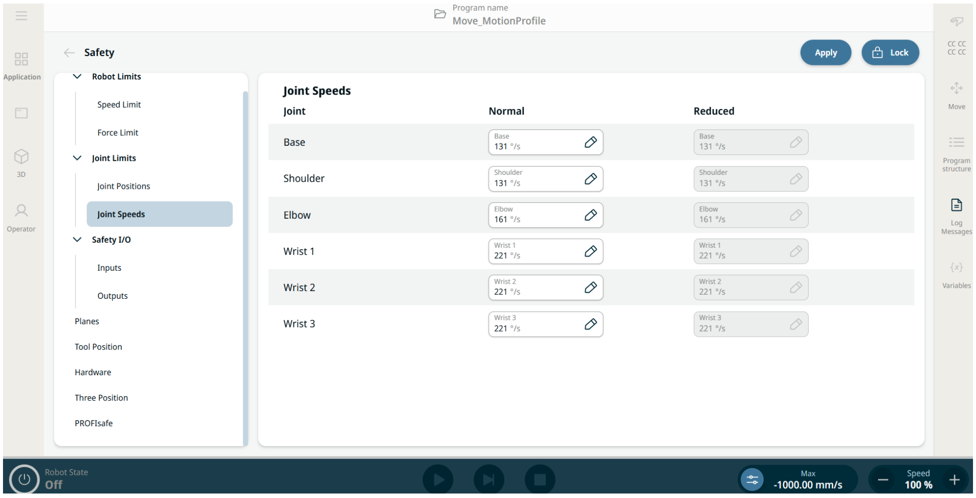
Joint Positions

Sets upper and lower limits for the allowed joint positions relative to the base of the robot.



Joint Speeds

Sets an upper limit for joint speed relative to the base of the robot.



20.8.4. Safety I/O

Description The I/O are divided between inputs and outputs and are paired up so that each function provides a Category 3 PLd capability.

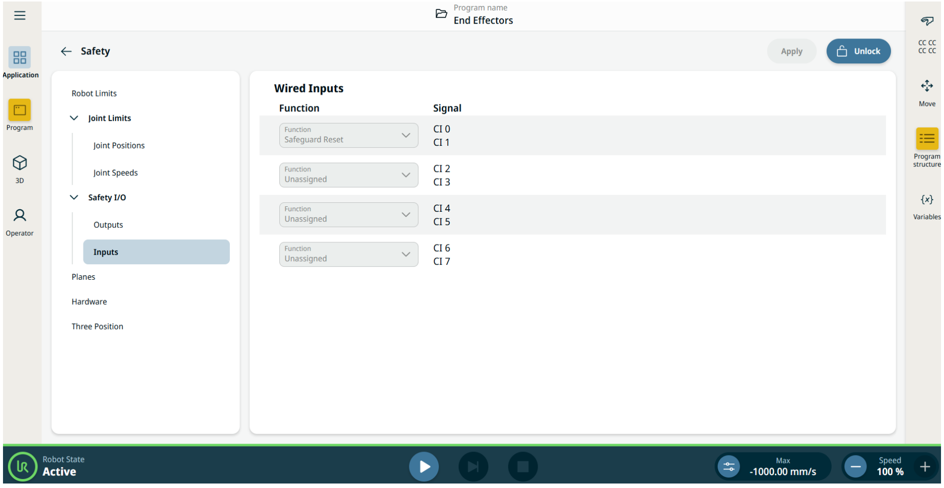
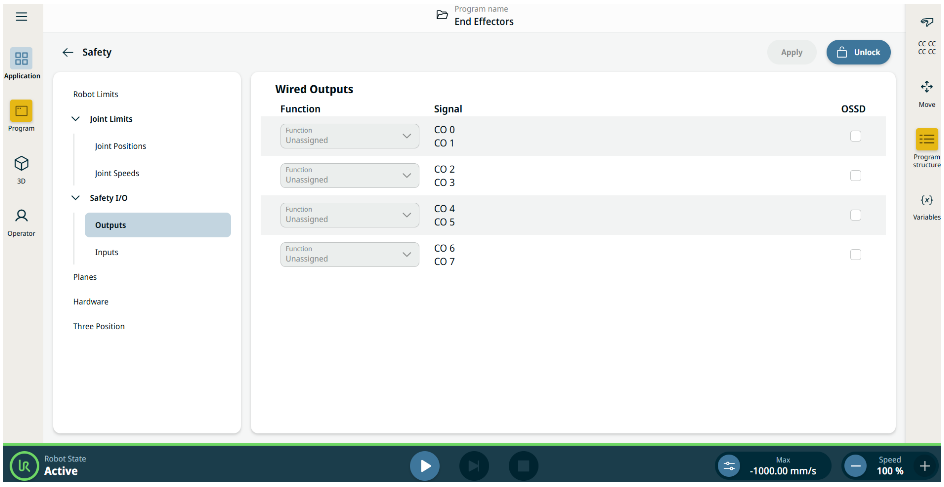


Figure 1.4: *PolyScope X screen displaying the Input signals.*



> **NOTICE**
> When starting programs from an I/O or fieldbus input, the robot can begin movement from the position it has, there will not be any manual movement to the first waypoint via PolyScope required.

Control Box Inputs

The inputs are described in the tables below:

Emergency Stop Button	Performs a Stop Category 1 (IEC 60204-1) informing other machines using the System Stop output if that output is defined. A stop is initiated in anything connected to the output.
Robot Emergency Stop	Performs a Stop Category 1 (IEC 60204-1) via Control Box input, informing other machines using the System Emergency Stop Output if that output is defined.
External Emergency Stop	Performs a Stop Category 1 (IEC 60204-1) on robot only.
Reduced	All safety limits can be applied while the robot is using a **Normal** configuration, or a **Reduced** configuration. When configured, a low signal sent to the inputs causes the safety system to transition to the reduced configuration. The robot arm decelerates to satisfy the reduced parameters. The safety system guarantees the robot is within reduced limits less than 0.5s after the input is triggered. If the robot arm continues to violate any of the reduced limits, a Stop Category 0 is triggered. Trigger planes can also cause a transition to the reduced configuration. The safety system transitions to the normal configuration in the same way.

Control Box Inputs

The inputs are described in the table below:

Operational Mode	When an external mode selection is used it switches between **Automatic Mode** and **Manual Mode**. The robot is in Automatic mode when input is *low* and Manual mode when the input is *high*.
Safeguard Reset	Returns from the Safeguard Stop state, when a rising edge on the Safeguard Reset input occurs. When a Safeguard Stop occurs, this input ensures that the Safeguard Stop state continues until a reset is triggered.
Safeguard	A stop triggered by a safeguard input. Performs a Stop Category 2 (IEC 60204-1) in all modes, when triggered by a Safeguard.
Automatic Mode Safeguard Stop	Performs a Stop Category 2 (IEC 60204-1) in Automatic mode ONLY. Automatic Mode Safeguard Stop can only be selected when a Three-Position Enabling Device is configured and installed.
Automatic Mode Safeguard Reset	Returns from the Automatic Mode Safeguard Stop state when a rising edge on the Automatic Mode Safeguard Reset input occurs.
Freedrive on robot	You can configure the Freedrive input to enable and use Freedrive without pressing the Freedrive button on a standard TP, or without having to press-and-hold any of the buttons on the 3PE TP in the light-press position.
3-Position Enabling Device	In Manual Mode, an external 3-Position Enabling Device must be pressed and held in the center-on position to move the robot. If you are using a built-in 3-Position Enabling Device, the button must be pressed and held in the mid position to move the robot.

WARNING

When the default Safeguard Reset is disabled, an automatic reset happens when the safeguard no longer triggers a stop.

This can happen if a person passes though the field of the safeguard.

If a person is not detected by the safeguard and the person is exposed to hazards, automatic reset is forbidden by standards.

- Use the external reset to ensure resetting only when a person is not exposed to hazards.

WARNING

When Automatic Mode Safeguard stop is enabled, a safeguard Stop is not triggered in Manual Mode.

Control Box Outputs

All safety outputs go low in the event of a safety system violation or fault. This means the System Stop output initiates a stop even when an E-stop is not triggered.

You can use the following Safety functions output signals. All signals return to low when the state which triggered the high signal has ended:

[1]System Stop	Signal is *Low* when the safety system has been triggered into a stopped state including by the Robot Emergency Stop input or the Emergency Stop Button. To avoid deadlocks, if the Emergency Stopped state is triggered by the System Stop input, low signal will not be given.
Robot Moving	Signal is *Low* if the robot is moving, otherwise high.
Robot Not Stopping	Signal is *High* when the robot is stopped or in the process of stopping due to an emergency stop or safeguard stop. Otherwise it will be logic low.
Reduced	Signal is *Low* when reduced parameters are active or if the safety input is configured with a reduced input and the signal is currently low. Otherwise the signal is high.
Not Reduced	This is the inverse of Reduced, defined above.
Safe Home	Signal is *High* if the robot arm is stopped and is located in the configured Safe Home Position. Otherwise, the signal is *Low*. This is often used when UR robots are integrated with mobile robots.
3-Position Enabling Stopped	Signal is low when a three position stop is active, high otherwise.
Not 3-Position Enabling Stopped	Signal is low when a three position stop is inactive, high otherwise.

NOTICE

Any external machinery receiving its Emergency Stop state from the robot through the System Stop output must comply with ISO 13850. This is particularly necessary in setups where the Robot Emergency Stop input is connected to an external Emergency Stop device. In such cases, the System Stop output becomes high when the external Emergency Stop device is released. This implies that the emergency stop state at the external machinery will be reset with no manual action needed from the robot’s operator. Hence, to comply with safety standards, the external machinery must require manual action in order to resume.

20.8.5. Safe Home

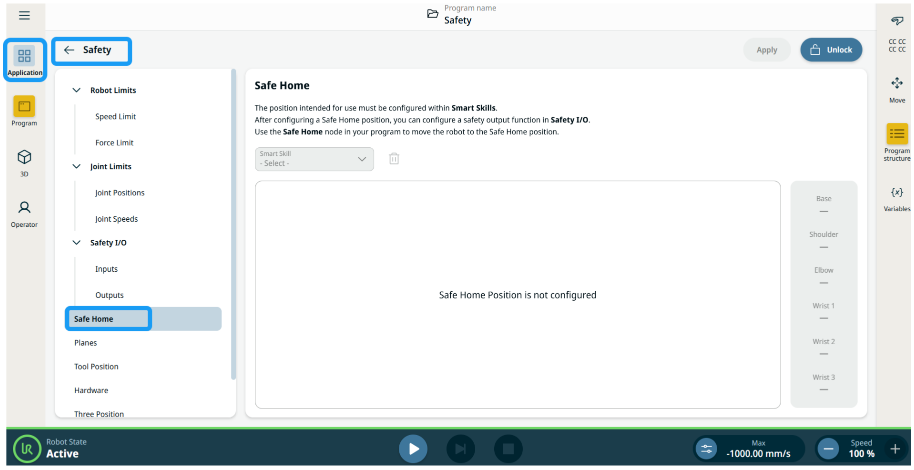
[1]System Stop was previously known as "System Emergency Stop" for Universal Robots robots. PolyScope can display "System Emergency Stop".

Description

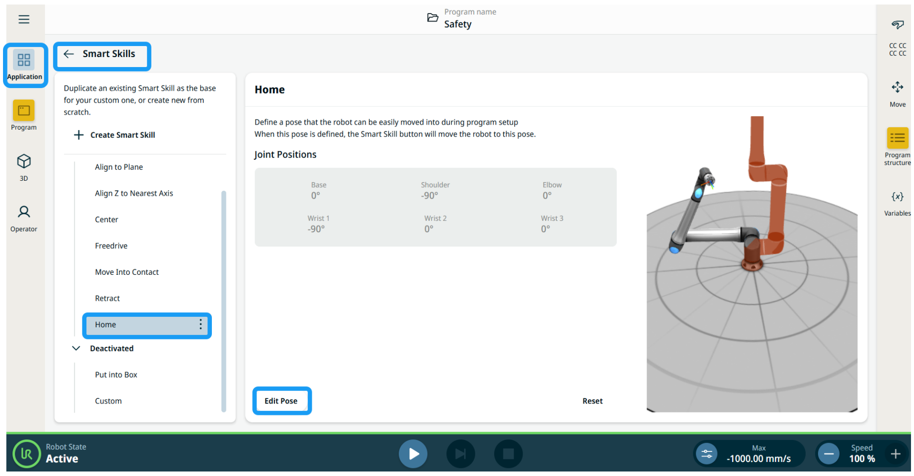
Safe Home is a PolyScope X feature that enables users to define a safety-related output that activates when the robot reaches the configured safe home position. The user chooses **Home** in the dropdown of available smart skill positions, which can be assigned as the reference for the Safe Home position.

Access Safe Home

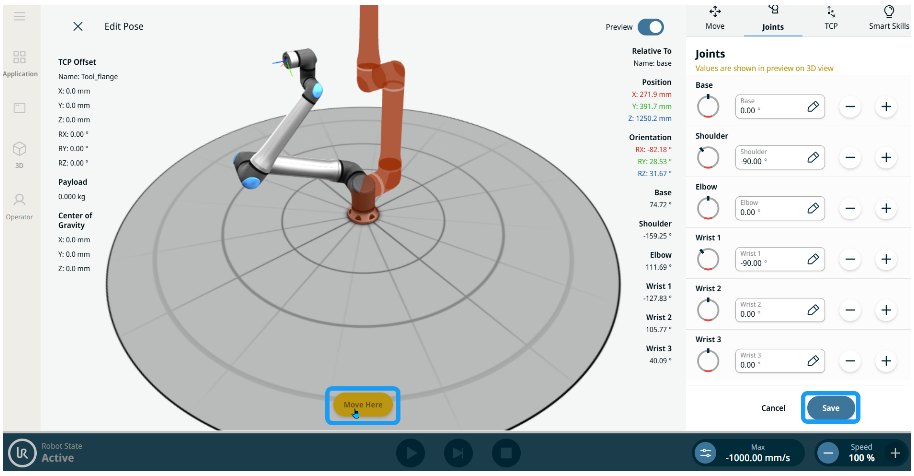
1. Tap the 20 Application Tab on page 218 and choose 20.8 Safety on page 264. In the left panel, choose **Safe Home**.



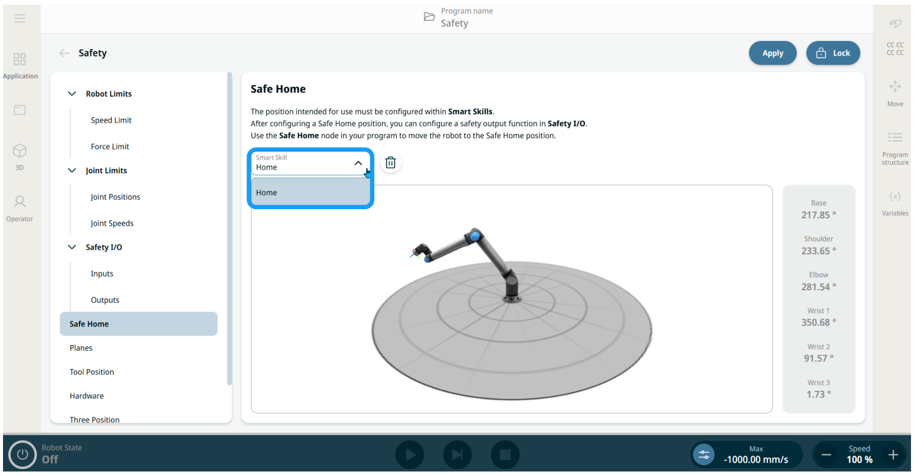
2. To configure the robot position, go to 20.9 Smart Skills Application on page 294 and choose **Home**.
3. Tap the **Edit Pose** button, and the 3D view of the robot arm appears.



4. Jog the robot to the intended position using the **Move**, **Joints**, **TCP** buttons.
5. Long press the yellow **Move Here** button until the previewed image is positioned to the new configuration.
6. Tap **Save** to finalize the configuration.



7. Go back to **Safe Home** in 20.8 Safety on page 264 and enter the safety password.
8. In the activated **Smart Skill** field, select **Home**, and the configured robot position appears in the main screen.
9. Tap **Apply** and choose **Apply and restart**. Then tap **Confirm Configuration**.

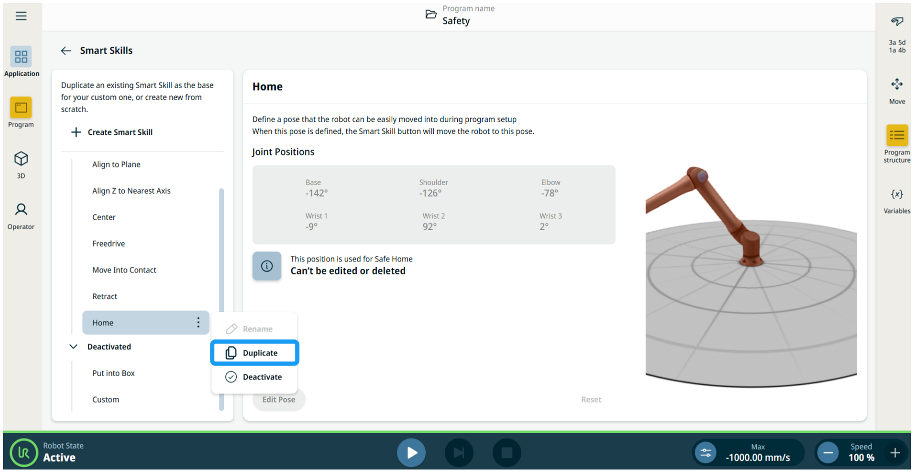


NOTICE

If the position smart skill is selected as the position reference for Safe Home:

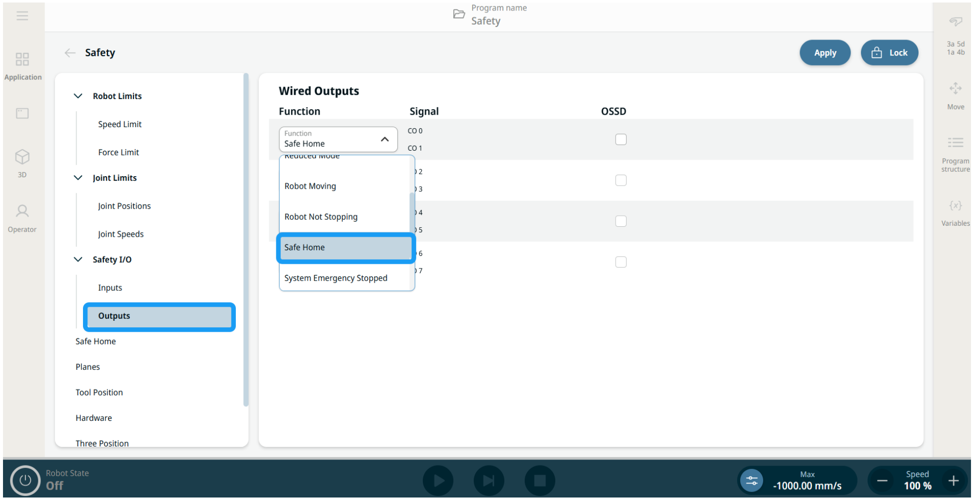
- A static message box is shown on the position smart skill page.
- A configured smart skill configured as Safe Home cannot be renamed, edited, or deleted unless the user deletes or unassigns the Safe Home from Safety Safe Home screen.

10. In the Smart Skills screen, the Home position can be duplicated and edited. All the newly created Smart Skills are also available to be configured as Safe Home.

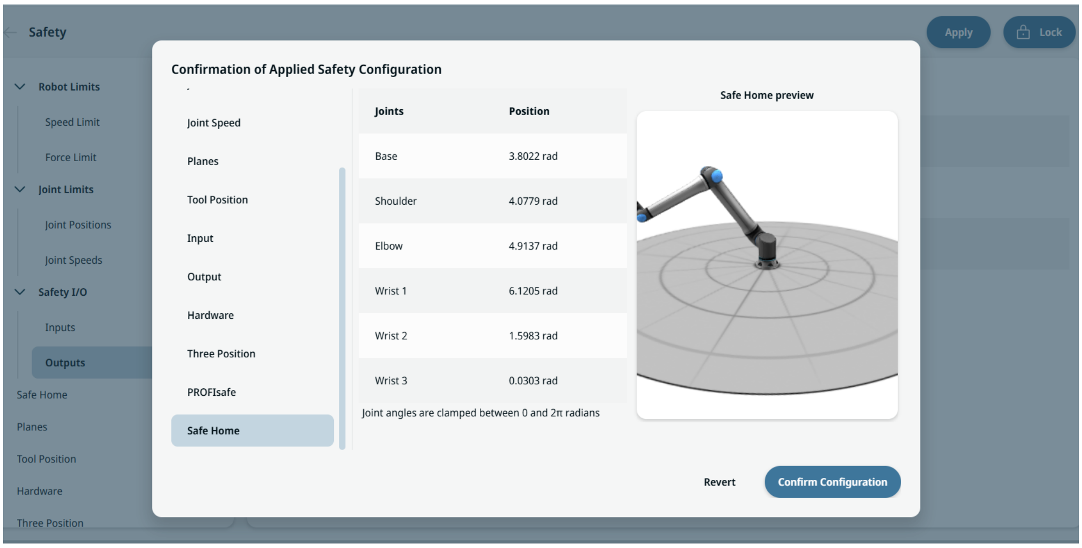


Safe Home function in outputs

If the **Home** position is assigned to a position smart skill in **Safety**, the safety output function dropdown contains **Safe Home** as a new function assignment.
Choose Safe Home in the **Function** fields in **Outputs** of Safety I/O.

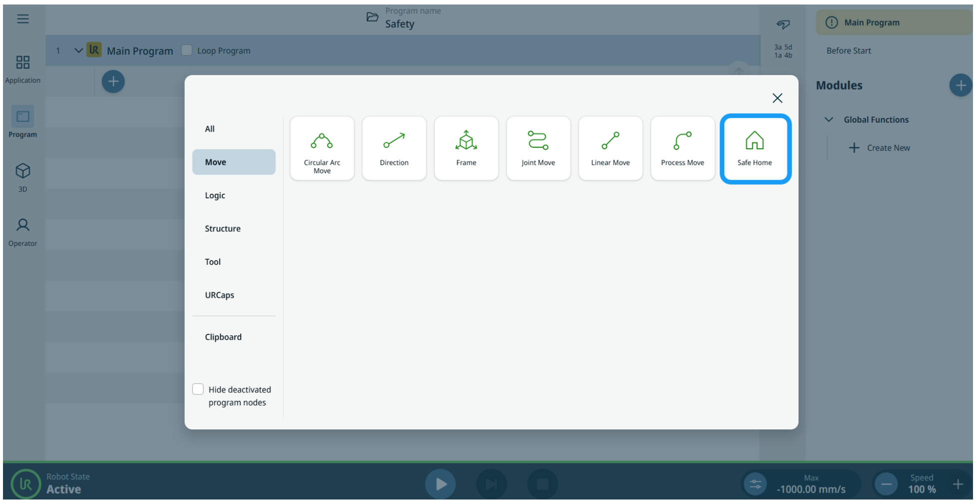


Safe Home can be seen in the table of function assignment on the **Confirmation of Applied Safety Configuration** overview dialog.



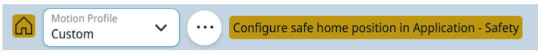
Safe Home as a program node

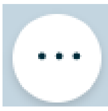
The **Safe Home** program node is located under the **Move nodes** category within the 16.3 Command Nodes on page 117. This node is used to command the robot to move to its predefined safe home position, as configured in the safety application settings for safe home.

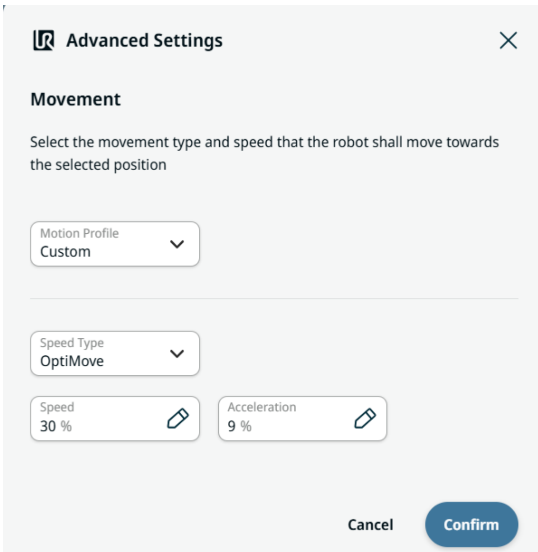


NOTICE

Adding the node without a safe home position configured in safety will mark the node as yellow.



The Safe Home command node has motion profiles options and other advanced settings when the ellipsis icon  is tapped.



Advanced Settings

Users can create movement profiles in the Advanced Settings using the two tabs and two fields:

- Motion Profile
- Speed Type
- Speed tab
- Acceleration tab

Motion Profile is a feature enabling users to set predefined motion profiles in move nodes: Joint Move, Linear Move, Circular Arc Move, Process Move, and Direction. **Custom** Motion Profile enables users to define movement data in move node.
In the **Speed Type** field, **OptiMove** is the default option. Choose **Classic** movement if you want to input speed and acceleration in degrees per second or mm per second. OptiMove settings are specified in percent to simplify the usage and setup.
Users can configure the percentage of preferred speed in the **Speed tab**, which sets the target travel speed as a percentage of the robot's maximum speed capability.
Users can configure the percentage of preferred acceleration in the **Acceleration tab**, which sets the target torque limits during acceleration and deceleration as a percentage of the robot's maximum power.

20.8.6. Planes

Description Safety planes restrict robot workspace, the tool, and the elbow.

> **WARNING**
>
> Defining safety planes only limits the defined Tool spheres and elbow, not the overall limit for the robot arm.
>
> Defining safety planes does not guarantee that other parts of the robot arm will obey this type of restriction.

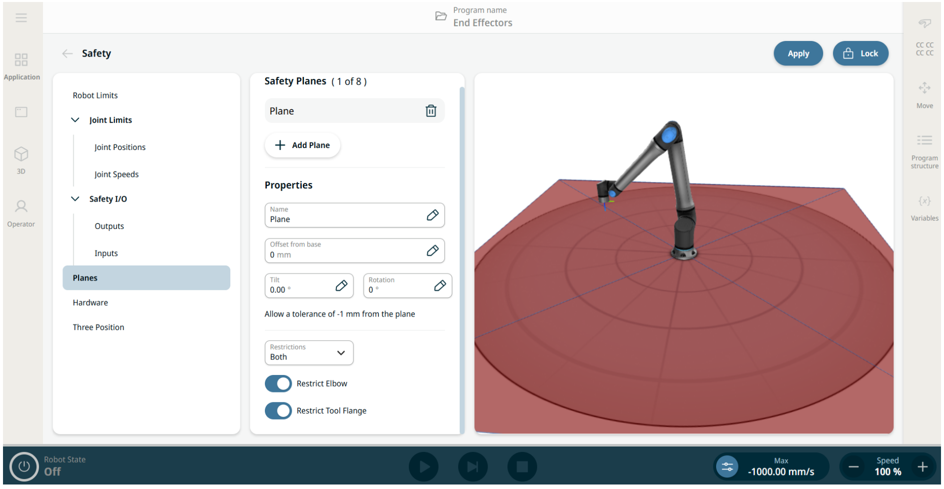


Figure 1.5: *PolyScope X screen displaying safety planes.*

Configuring a Safety Plane

You can configure safety planes with the properties listed below:

- **Name**. This is the name used to identify the safety plane.
- **Offset from base**. This is the height of the plane from the base, measured in the Y direction.
- **Tilt**. This is the tilt of the plane, measured from the power cord.
- **Rotation**. This is the rotation of the plane, measured clockwise.

You can configure each plane with the restrictions listed below:

- **Normal**. When the safety system is in Normal mode, a normal plane is active and it acts as a strict limit on the position.
- **Reduced**. When the safety system is in Reduced mode, a reduced mode plane is active and it acts as a strict limit on the position.
- **Both**. When the safety system is either in Normal or Reduced mode, a normal and reduced mode plane is active and acts as a strict limit on the position.
- **Trigger Reduced Mode**. The safety plane causes the safety system to switch to Reduced mode if the robot Tool or Elbow is positioned beyond it.

Elbow Joint Restriction

The feature is enabled by default.
You can use Restrict Elbow to prevent robot elbow joint from passing through any of your defined planes.
Disable Restrict Elbow for elbow to pass through planes.

Tool Flange Restriction

Restricting the tool flange prevents the tool flange and the attached tool from crossing a safety plane. When you restrict the tool flange, the unrestricted area is the area inside of the safety plane, where the tool flange can operate normally.
The tool flange cannot cross the restricted area, outside of the safety plane.
Removing the restriction allows the tool flange to go beyond the safety plane, to the restricted area, while the attached tool remains inside of the safety plane.
You can remove the tool flange restriction when working with a large tool off-set. This will allow extra distance for the tool to move.
Restricting the tool flange requires the creation of a plane feature. The plane feature is used to set up a safety plane later in the safety settings.

20.8.7. Tool Position Restriction

Description

The Tool Position screen enables users a more controlled restriction of tools and/or accessories placed on the end of the robot arm by letting you define tool positions with a radius that will interact with the safety planes by either collision detection with Tool Position and plane or enter reduced mode when tool enters plane.

Details

Tool Position has two key benefits:

- Supports two custom configurations to specify where to react to safety planes.
- Visualizes Tool Positions in 3D model.

> **NOTICE**
> You can define, configure, and manage up to two tool positions.

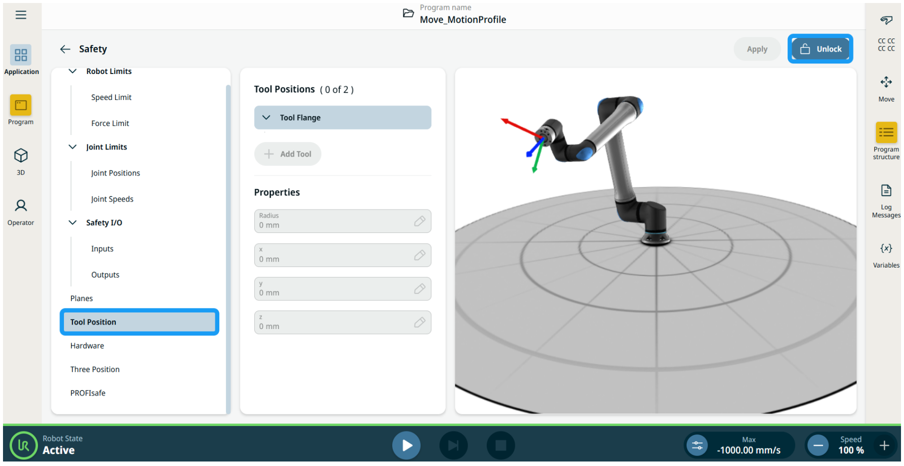
User defined tools

For the user defined tools, the user can change:

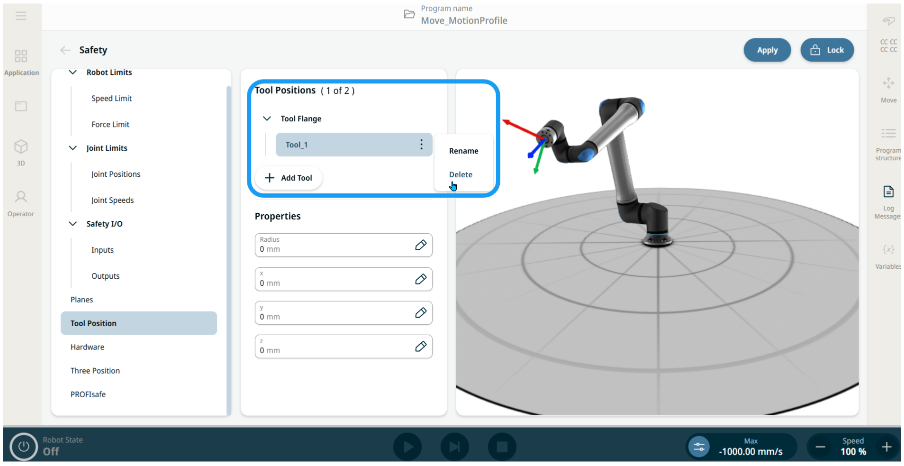
- Radius to change the radius of the tool sphere. The radius is considered when using safety planes.
- X, Y, Z positions to change the position of the tool with respect to the tool flange of the robot. The position is considered for the safety functions for tool speed, tool force, stopping distance, and safety planes.

To access Tool Position

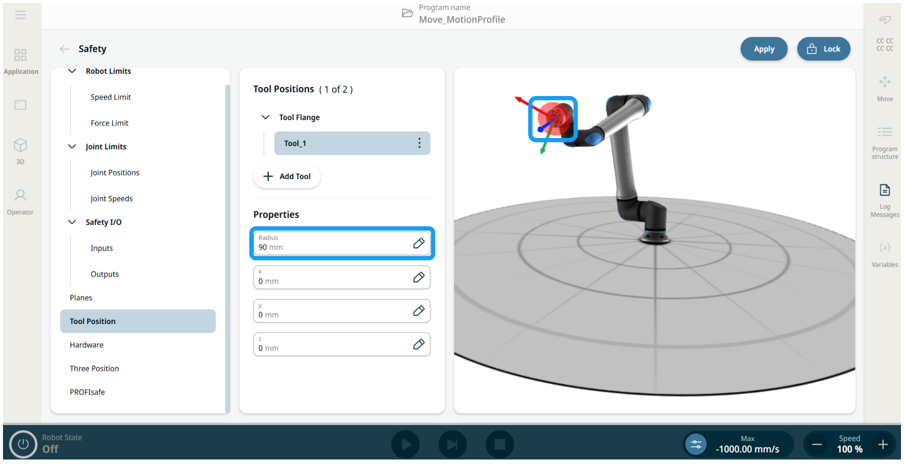
1. Navigate to **Safety** Application.
2. Tap **Tool Position** in the left panel. On the top right side of the main screen, tap **Unlock** to activate adding tools. Enter the safety password and **Confirm**.



3. In the central panel's **Tool Positions** column, tap **+ Add Tool**. The added tool, **Tool_1**, appears under the **Tool Flange tree**.
4. Tap the **kebab icon** of the added tool to rename it to something more identifiable. You can also delete it.



5. In the central panel's **Properties** column, you can find four editable fields for **radius, x, y, and z positions**. Tap the fields to change the radius and x ,y, z coordinates as needed. The sphere in the right panel updates live in the 3D model to assist with accurate placement.
6. Tap **Apply** in the top-right part of the main screen.



7. The robot will now interact with safety planes when the tool position spheres come into contact with them.

20.8.8. Hardware

Description

The **Hardware** section allows users to select the type of Teach Pendant (TP) connected to their robot. Available options are:

- **3PE Enabled**. 3-Position Enabling Teach Pendant (3PE TP)
- **None**. This allows the robot to be safely powered up without a Teach Pendant connected to the Control Box.

> **NOTICE**
>
> For safety reasons, the robot will not start if the selection does not reflect the connected hardware.

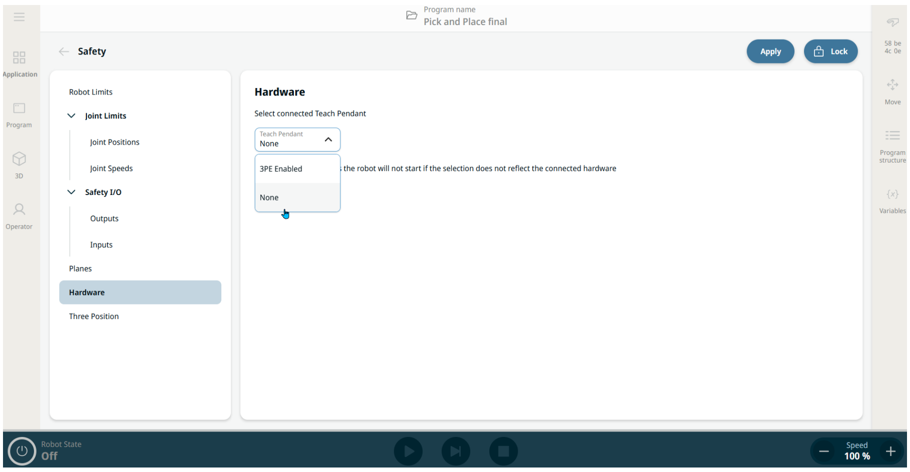
Purpose of None option

The **None** option supports setups where a Teach Pendant is not required or physically present, enabling more streamlined and flexible deployments, especially in automated or remote-controlled environments.
It gives users three key benefits:

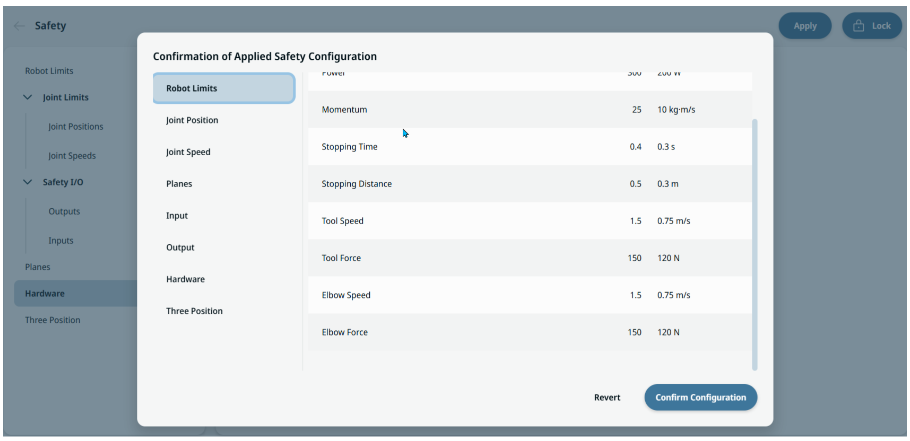
- Supports headless configurations where no Teach Pendant is used.
- Simplifies hardware requirements for minimal setups.
- Improves start-up flexibility by removing dependency on Teach Pendant presence.

To access None

1. Tap **Hardware** on the left panel of the Safety application screen.
2. Unlock the main screen in the topmost right part, enter your safety password, and **Confirm**.
3. On the central panel, tap the **Teach Pendant** field, and choose **None**.



4. Tap **Apply** beside the Lock button in the main screen.
5. Tap **Apply and restart** in the pop-up of Apply Safety Configuration.
6. A Confirmation of Applied Safety Configuration pop-up screen appears, and you can either **Revert** or **Confirm Configuration**.



20.8.9. Three Position

Description	Manual high speed allows both tool speed and elbow speed to temporarily exceed 250 mm/s. It is only available when your robot is on manual mode and a three-position enabling device is configured. If there is no interaction with the three-position enabling device in five minutes, increased values will be reset to 250 mm/s.
To access Three Position	1. On the left panel, tap **Three Position**. 2. Tap **Unlock** on the upper right side of the main screen. 3. Enter the safety password and **Confirm**. 4. On the central panel, slide on the button to **Allow manual high speed**. 5. Tap **Apply** on the upper right side of the main screen.

20.8.10. PROFIsafe

Description	The PROFIsafe network protocol (implemented as version 2.6.1) allows the robot to communicate with a safety PLC according to ISO 13849, Cat 3 PLd requirements. The robot transmits safety state information to a safety PLC and receives commands to reduce safety limits or trigger a safety-related function such as an emergency stop. The PROFIsafe interface provides a safe, network-based alternative to connecting wires to the safety IO pins of the robot control box. PROFIsafe is only available on robots that have an enabling license, which you can obtain by contacting your local sales representative. After you obtain the license, download it from myUR. For information about robot registration and license activation, see Robot Registration and URCap License files.

Advanced Options

A control message that the safety PLC sends to the robot contains the information shown in the following table.

Signal	Description
E-Stop by system	0: Asserts the system e-stop. 1: Clear system e-stop.
Safeguard stop	0: Asserts the safeguard stop. 1: Normal operation state. **Note**: Also refer to the “Reset safeguard stop” signal description.
Reset safeguard stop	Resets the safeguard stop state on a 0-to-1 transition when the “safeguard stop” signal is already set to 1.
Safeguard stop auto	0: Asserts safeguard stop if the robot is operating in Automatic mode. 1: Normal operation state. Safeguard stop auto shall only be used when a 3-Position Enabling (3PE) Device is configured. If no 3PE Device is configured, the safeguard stop auto acts as a normal safeguard stop input. **Note**: Also refer to the “Reset safeguard stop auto” signal description.
Reset safeguard stop auto	Resets the safeguard stop auto state on a 0-to-1 transition when the “safeguard stop auto” signal is already set to 1.
Reduced	0: Activates the Reduced safety limits. 1: Activates the “Normal mode” safety limits. The safety system guarantees the robot is within reduced limits less than 0.5s after the input is activated. If the robot arm continues to violate any of the reduced limits, a Stop Category 0 is triggered.
Operational mode	0: Activates the manual operational mode. 1: Activates the automatic operational mode. If the safety configuration "Operational mode selection via PROFIsafe" is disabled, this field shall be omitted from the PROFIsafe control message.

Advanced Options

A status message that the robot sends to the safety PLC contains the information shown in the following table.

Signal	Description
Stop, cat. 0	0: Robot is performing, or has completed, a safety stop of category 0; a hard stop by immediate removal of power to the arm and the motors. 1: Normal operation state.
Stop, cat. 1	0: Robot is performing, or has completed, a safety stop of category 1; a controlled stop after which the motors are left in a power-off state with brakes engaged. 1: Normal operation state.
Stop, cat. 2	0: Robot is performing, or has completed, a safety stop of category 2; a controlled stop after which the motors are left in a power-on state. 1: Normal operation state.
Violation	0: Robot is stopped because the safety system has failed to comply with the active safety limits defined. 1: Normal operation state.
Fault	0: Robot is stopped because of an unexpected exceptional error in the safety system. 1: Robot is not experiencing an unexpected exceptional error in the safety system.
E-stop by system	0: Robot is stopped because of one of the following conditions: • A safety PLC connected via PROFIsafe has asserted a system level e-stop. • An IMMI module connected to the control box has asserted a system level e-stop. • A unit connected to the system e-stop configurable safety input of the control box has asserted a system level e-stop. 1: Robot is not in system e-stop.
E-stop by robot	0: The robot is stopped because of one of the following conditions: • The e-stop button of the teach pendant is pressed. • An e-stop button connected to the robot e-stop non-configurable safety input of the control box is pressed. 1: Robot is not in e-stop by robot.

Advanced Options

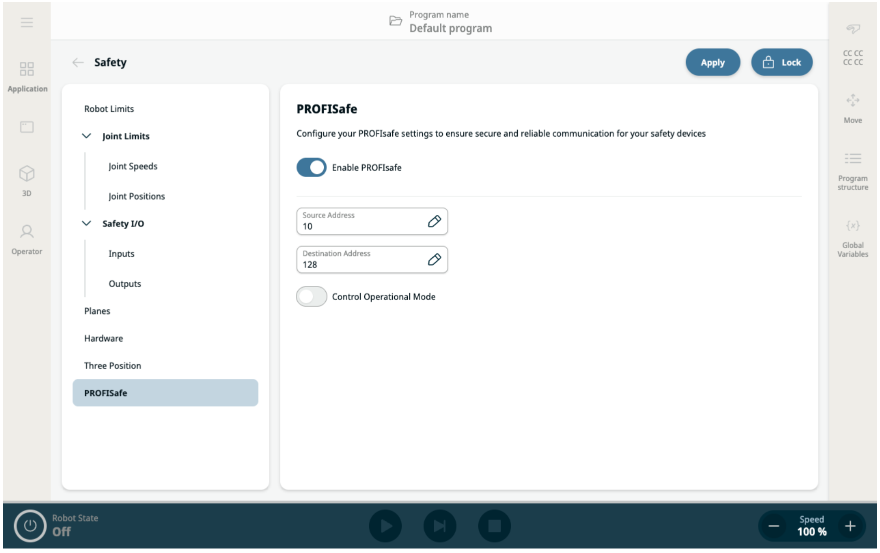
Signal	Description
Active limit set	The active set of safety limits. 0: Normal 1: Reduced 2: Recovery
Robot moving	0: Robot is moving. If any joint moves at a velocity of 0.02 rad/s or higher, the robot is considered in motion. 1: Robot is at standstill.
Safe home position	0: Robot is at rest (robot not moving), and in the position defined as the Safe Home Position. 1: Robot is not at rest, or not in the position defined as the Safe Home Position.

Advanced Options

Signal	Description
Safeguard stop	0: The robot is stopped due to one of the following conditions: • A safety PLC connected via PROFIsafe has asserted the safeguard stop. • A unit connected to the safeguard stop nonconfigurable input of the control box has asserted the safeguard stop. • A unit connected to the safeguard stop configurable safety input of the control box has asserted the safeguard stop. 1: The robot is not stopped due to a safeguard stop. **Note**: Also refer to the “Reset safeguard stop” signal description. PROFIsafe enforces the use of the safeguard reset functionality.
Safeguard stop auto	0: The robot is stopped because it is operating in Automatic mode and one of the following conditions applies: • A safety PLC connected via PROFIsafe has asserted safeguard stop auto. • A unit connected to a safeguard stop auto configurable safety input of the control box has asserted safeguard stop auto. 1: The robot is not stopped due to safeguard stop auto. **Note**: Also refer to the “Reset safeguard stop auto” signal description. PROFIsafe enforces the use of the safeguard reset functionality.
3PE stop	0: The robot is stopped because it is operating in Manual mode and one of the following conditions applies: • Any 3PE is pressed to the middle position, and Freedrive input is active. • Not all 3PE devices are pressed to the middle position. 1: Robot is not stopped because of a 3-position enabling device.
Operational mode	Indication of the active operational mode of the robot. 0: Disabled 1: Automatic 2: Manual
Reduced	0: Reduced safety limits are active. 1: Normal safety limits are active.

Configuring PROFIsafe

1. On the Safety application screen, tap **PROFIsafe** on the left panel.
2. Tap **Unlock** on the top right main screen to enable PROFIsafe. Enter the safety password and **Confirm**.



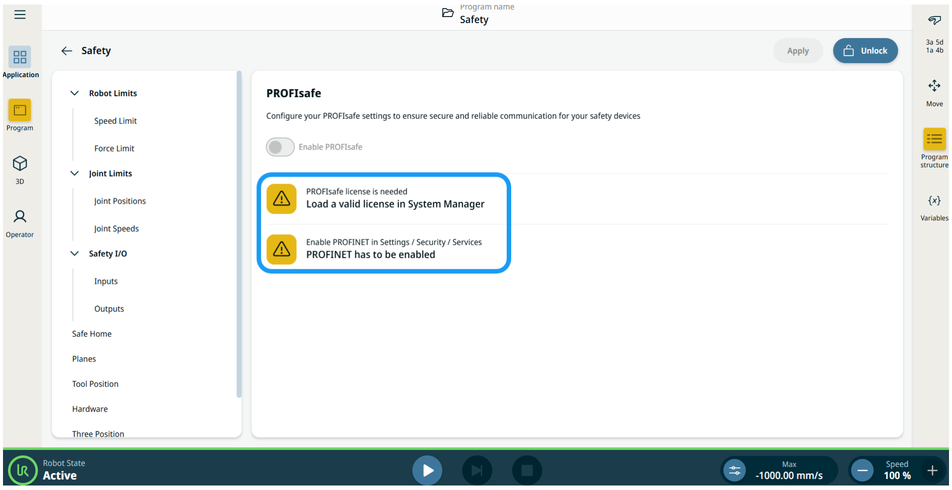
The right panel shows two fields and two buttons for configuring PROFIsafe:

- Enable PROFIsafe button
- Source Address field
- Destination Address field
- Control Operational Mode

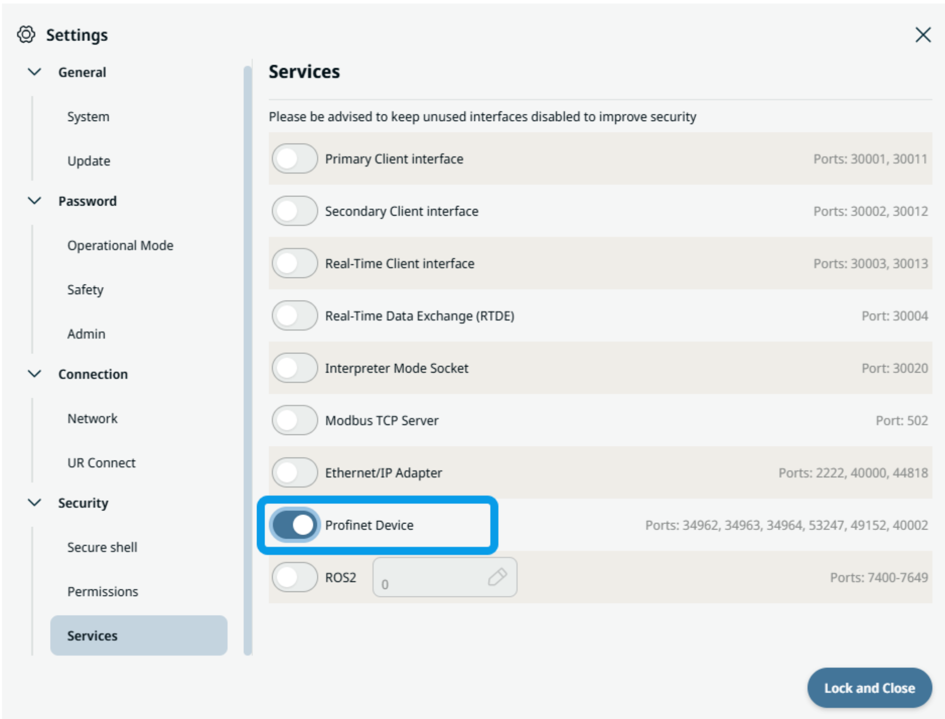
3. Slide right the **Enable PROFIsafe** button.
4. Tap the **Source Address** and **Destination Address** fields to specify the addresses to be used by the robot and the safety PLC in identifying each other.
5. By tapping **Control Operational Mode**, you have the option to enable the PROFIsafe PLC to control the robot operational mode.

NOTICE

In order to configure and use PROFIsafe, **PROFINET Device** must be enabled in the security services settings menu.



Read for details and interface location PROFINET Device.



20.9. Smart Skills Application

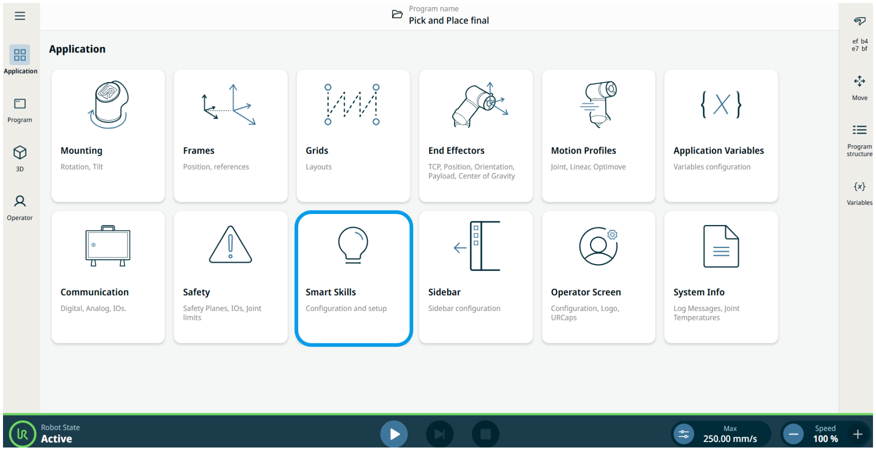
Description

All Smart Skills are parametrized, which enables you to configure and set up the Smart Skills in the Application tab. The Smart Skills interface gives you the following benefits:

- A clear and accessible call-to-action button for creating new Smart Skills.
- Eliminates the need to duplicate existing skills as a starting point.
- Simplifies and accelerates the creation process.
- Improves clarity and usability within the Smart Skills configuration interface.

Using the Smart Skills application functionality

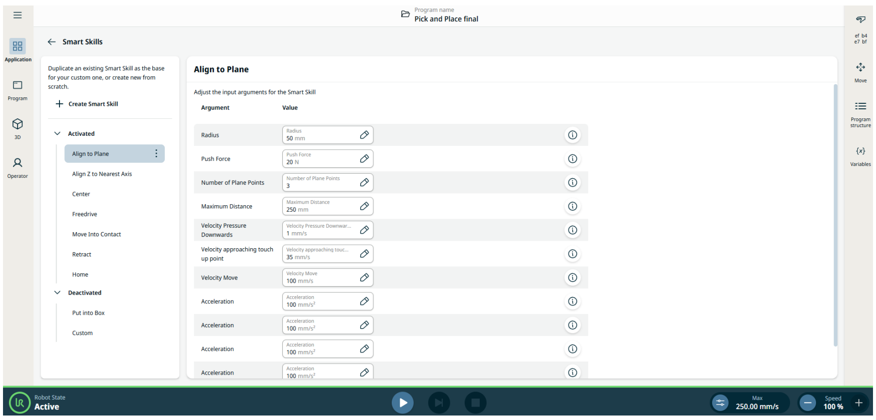
1. Go to the application nodes screen. See Application Tab.
2. Tap the **Smart Skills** icon.



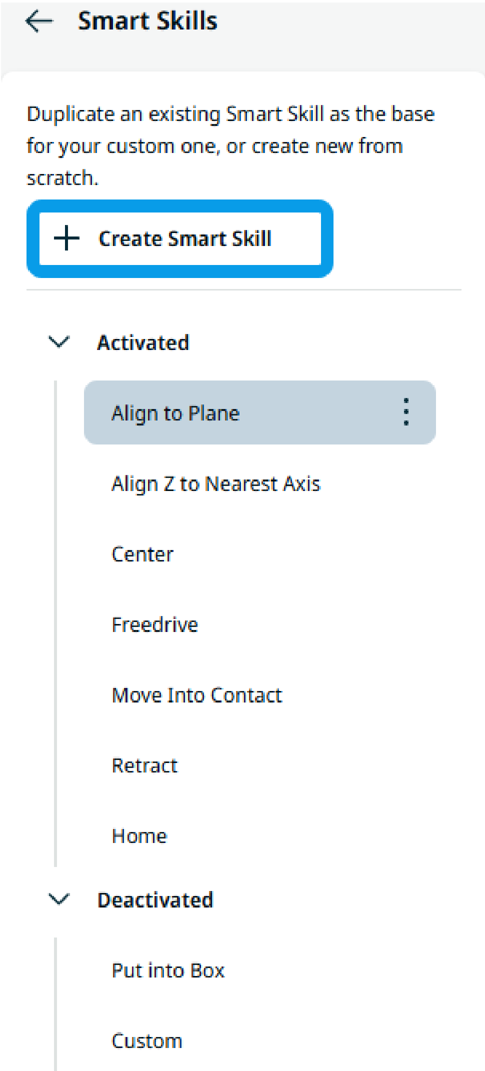
A screen appears with two panels:

Left panel contains the list of smart skills. Tap the kebab icon of each smart skill to activate or deactivate it. When activated, that smart skill appears in the Smart Skills drawer in the multitasking screen and disappears when deactivated.

Right panel contains the editable fields to adjust the input arguments for each smart skill.



The process of creating custom Smart Skills is streamlined for ease of use. Instead of duplicating existing skills, tap **+ Create Smart Skill** button to start from scratch, found at the left panel before the list of Smart Skills. This makes it easier and more intuitive for users to build tailored Smart Skills, enhancing productivity and reducing setup time.



Smart Skill Input Argument

Each Smart Skill has a couple of parameters it uses when it executes. These can be changed to fit the application that you are running, whether or not it fits the default values.

These are the input arguments of the smart skills where you can change the value:

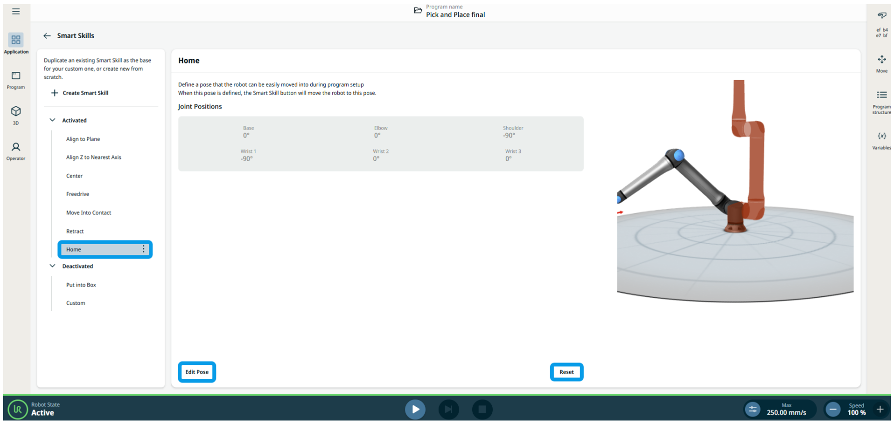
- Align to Plane
  - **Radius**. Radius of the circle within the plane that will be touched up.
  - **Push Force**. How hard the robot pushes against the plane.
  - **Number of Plane Points**. Number of points the robot uses to compute the plane.
  - **Maximum Distance**. The maximum distance the robot searches.
  - **Velocity Pressure Downwards**. Velocity when pressing downwards.
  - **Velocity Approaching Touch Up Point**. Velocity used when approaching the touch-up point.
  - **Velocity Move**. Velocity used in freespace.
  - **Acceleration**. Acceleration of the robot.
- Align Z to Nearest Axis

  Not configurable
- Center
  - **Force**. Force the robot uses to determine if a contact has been achieved.
  - **Velocity**. Velocity in freespace.
  - **Acceleration**. Acceleration in freespace.
  - **Maximum Radius**. The maximum search radius.
  - **Number of Fingers**. The number of fingers that the gripper has.
- Freedrive

  Not configurable
- Move Into Contact
  - **Force**. Force the robot uses to determine if a contact has been achieved.
  - **Velocity**. Velocity in freespace.
  - **Acceleration**. Acceleration in freespace.
  - **Maximum Distance**. The maximum distance the robot searches.
  - **Retract Distance**. Retract distance after a contact has been found.
- Retract
  - **Distance**. Retraction distance.
  - **Acceleration**. Acceleration used by the robot.
  - **Velocity**. Velocity used by the robot.

- Home

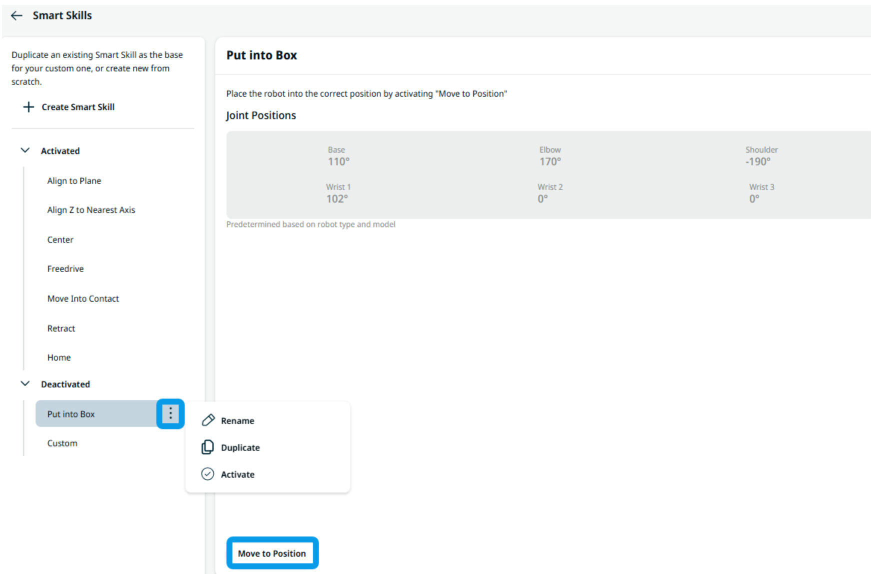
  This enables you to define a posture to easily move the robot to its predefined home position. Tap **Edit Posture**, and you will be directed to the 3D viewer main screen. Tap **Reset**, and a dialogue box appears asking you to confirm or cancel. When the position is reset, the previous values will be deleted.



Deactivated Smart Skills by default

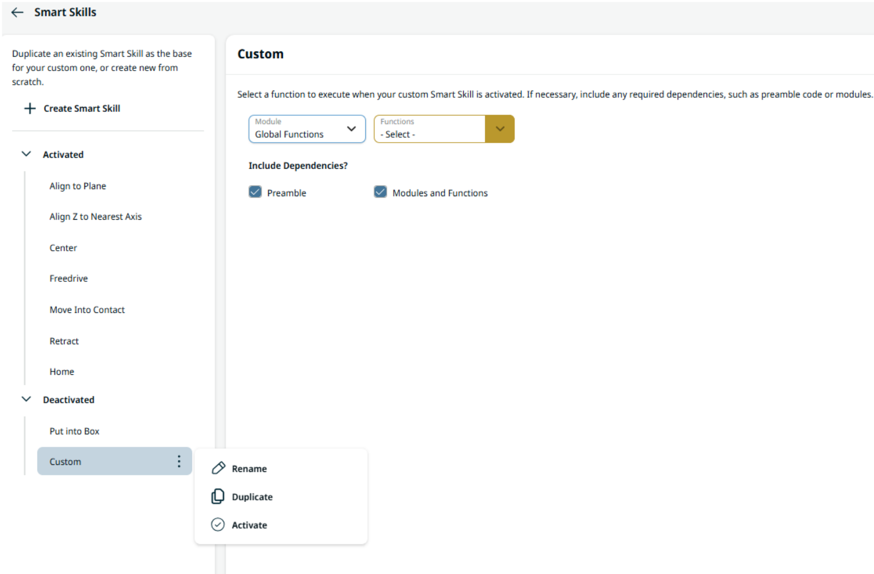
- Put Into Box

  This smart skill enables you to put the robot in its default transportation position. Tap **Move to Position** button below to place the robot into the correct position. Its **Preset Joint Position** is deactivated as it is predetermined by the robot type and model.



- Custom

  You can duplicate and rename this using the kebab icon. Custom is an easy way to execute a function of a module. It is then shown in the Smart Skills icon.



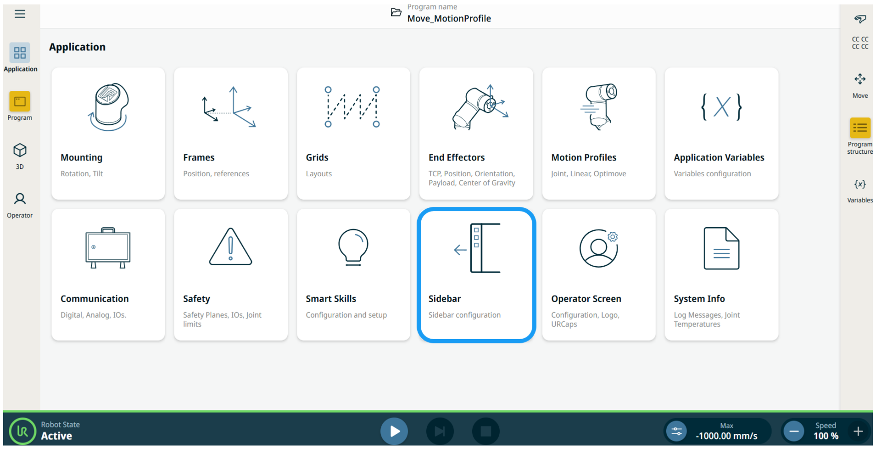
20.10. Sidebar in Application Tab

Description

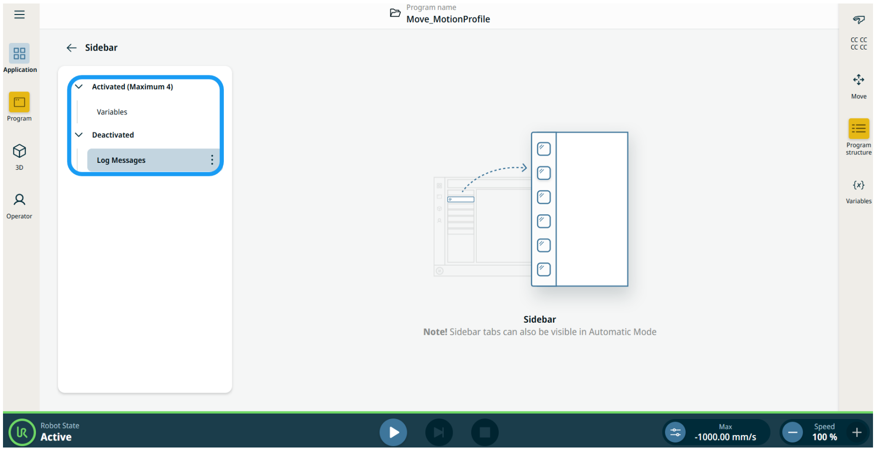
The Sidebar in the Application Tab enables users to add tabs in addition to the four main menus in the multitask screen.

Using the Sidebar functionality

1. Go to the application nodes screen. See Application Tab.
2. Tap the **Sidebar** icon.

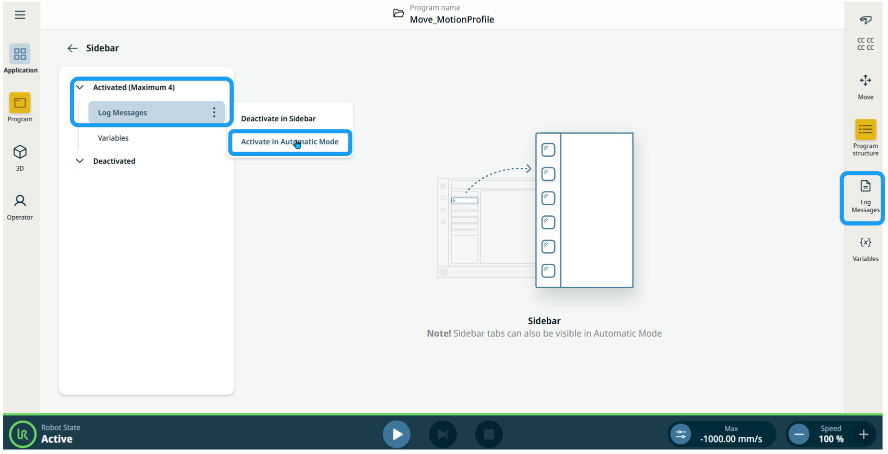


The main screen shows the **Activated** and **Deactivated** sidebar tree.

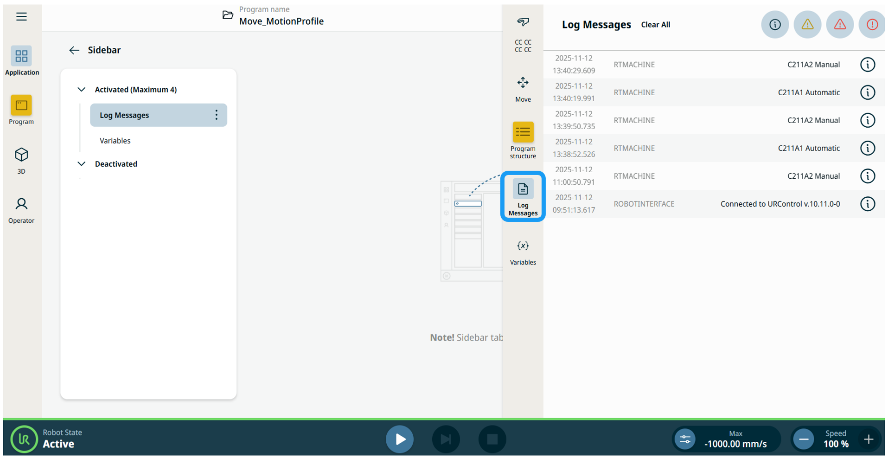


Log Messages

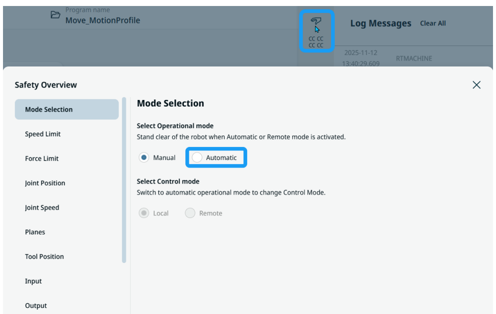
1. Tap the three vertical dots (kebab icon) of Log Messages and choose **Activate in Sidebar** and **Activate in Automatic Mode**. Log Messages is then transferred to the Activated sidebar tree.
2. In the sidebar, you can see **Log Messages** is inserted.



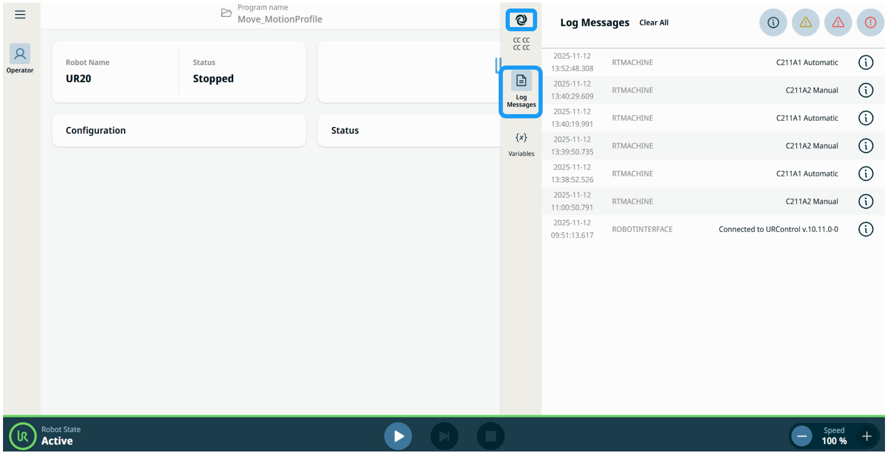
3. Tap Log Messages in the sidebar. The multitask screen expands to show the log messages information.



4. To switch to automatic mode, tap the Safety Checksum icon.



5. Choose **Automatic** in the **Mode Selection** screen. Enter the operational mode password to unlock and confirm.
6. The interface in automatic mode appears, signified by the swirling icon in the safety checksum, which still shows the Log Messages in the sidebar.



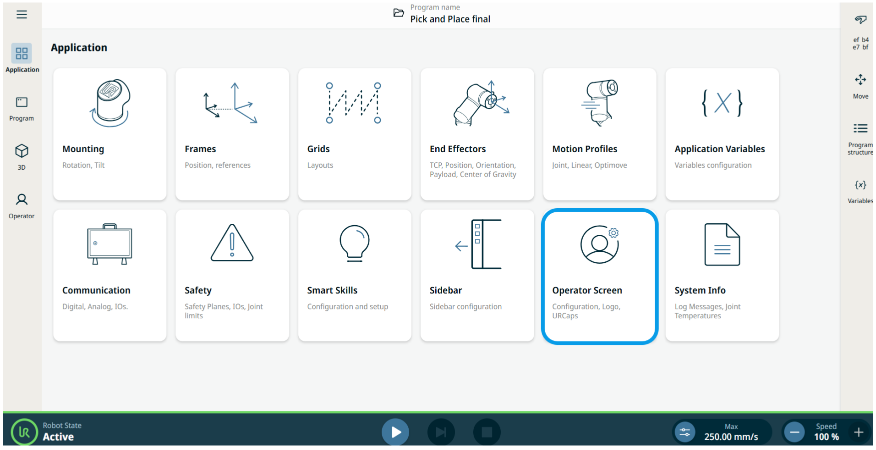
20.11. Operator Screen Application

Description

The **Operator Screen** application enables you to make changes in the contents of the Operator, located in the left toolbar. See Operator Screen.

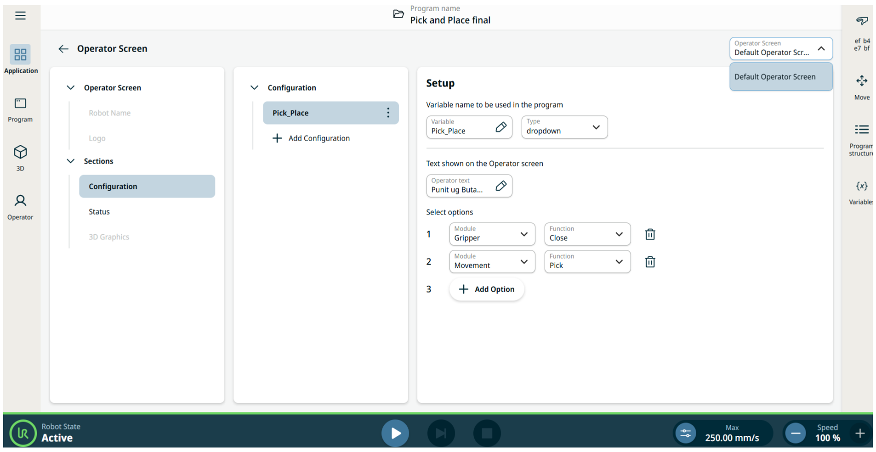
Using the Operator Screen application functionality

1. Go to the application nodes screen. See Application Tab.
2. Tap the **Operator Screen** icon.



You see the Operator Screen in the main screen, which is divided into two panels and the Operator Screen field.

The left panel contains the **Operator Screen** and **Sections** tree. The center panel contains the **Configuration** tree. The Operator Screen field contains only **Default Operator Screen** option.



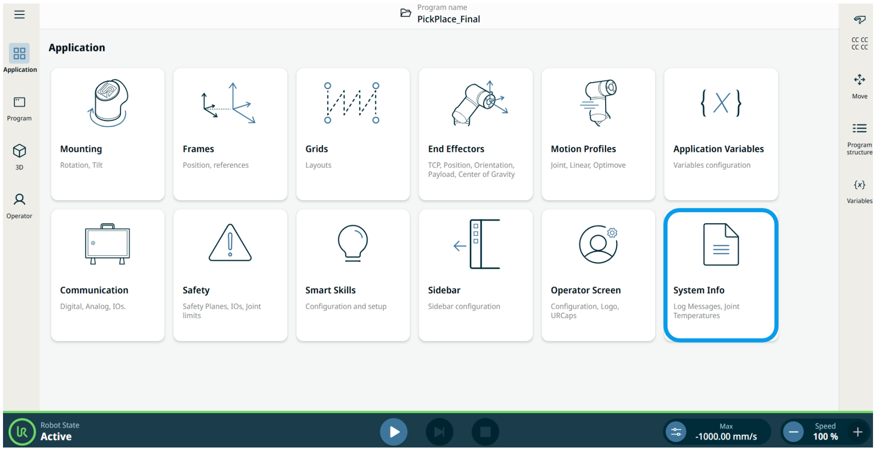
20.12. System Info

Description

The **System Info** screen lists all log messages from the system, enables exporting support files, and checks if the robot join temperature and current reach an unexpected level.

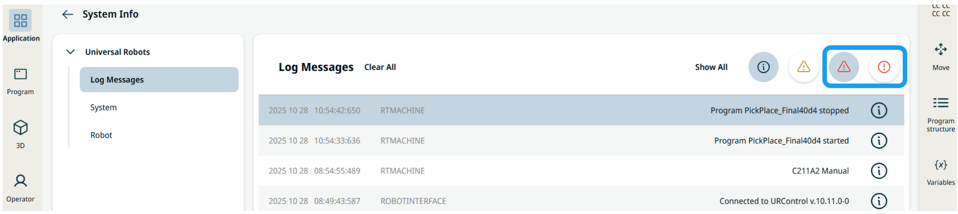
Using the System Info functionality

1. Go to the Application menu screen. See Application Tab.
2. Tap the **System Info** icon.

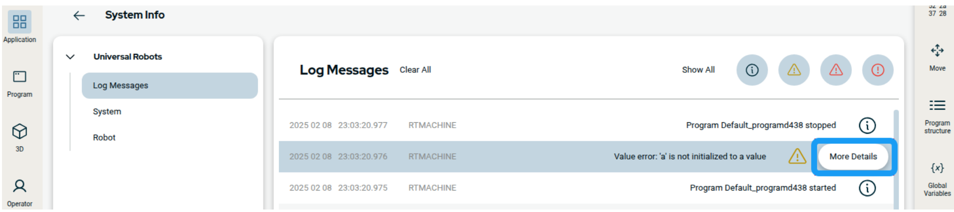


Log Messages

The **Log Messages** page lists all log messages from the system. The page has four buttons at the top of the messages list. The two red buttons allow for filtering on messages for faults and violations independently. The rightmost filter button shows/hides violations, whereas the filter button next to it shows/hides faults.

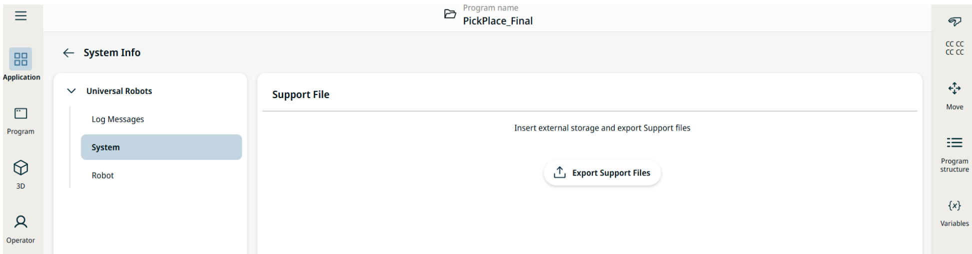


For some log messages, it is possible to view additional information by pressing the **More Details** button that appears when the message is selected in the list.

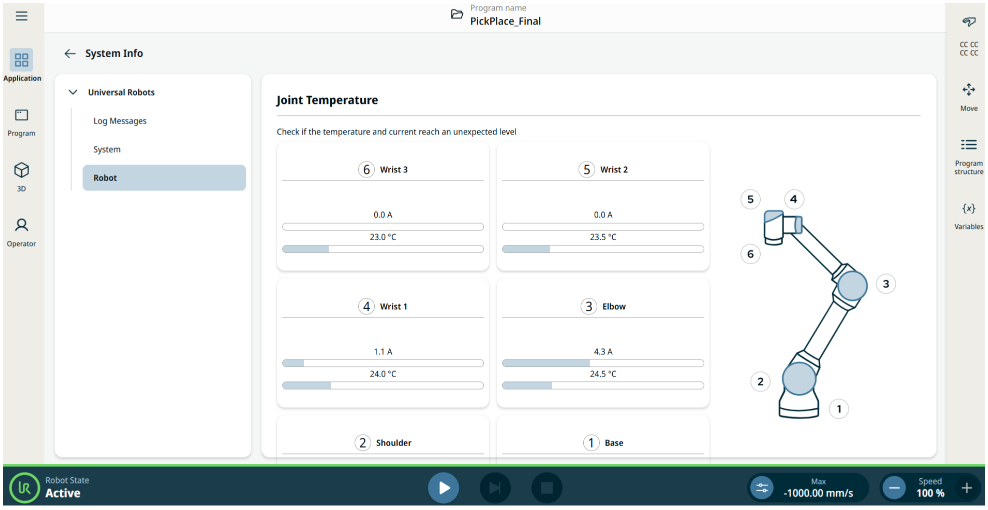


A dialog box displays with more details.

System The **System** page enables you to insert external storage and export support files.



Robot The robot **Joint Temperature** checks if the robot joint temperature and current reach an unexpected level.



21. Operator Screen Configuration

Description

Operator Screen configuration is a user action to personalize the main screen. This gives the advantage of seeing the name and status of your created program. A further benefit is the guarantee that it is you who legitimately creates a program. See Operator Screen Application.

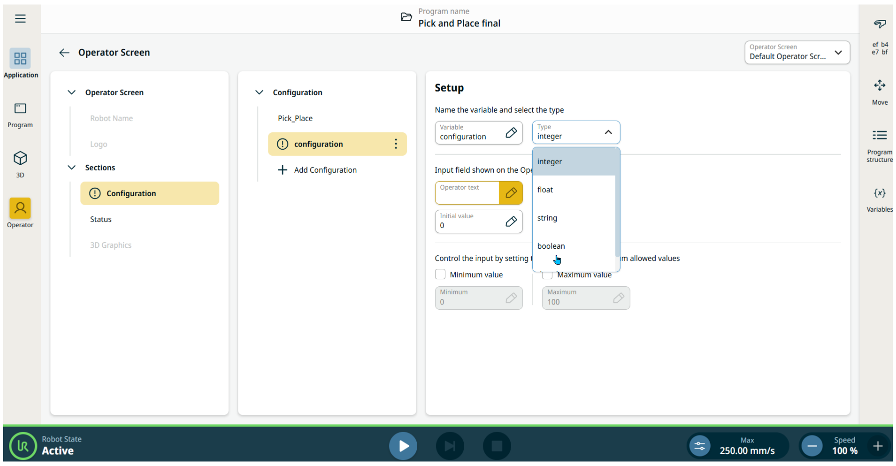
To configure Operator Screen

1. Go to the Application nodes screen. See Application Tab and Operator Screen Application.
2. On the center panel of the Operator Screen, tap **Add Configuration** under the Configuration tree.

   A **Setup** right panel is inserted, which contains the following components:

   - Variable field
   - Type field. This determines what configuration will be added.
   - Operator text field
   - Initial Value field
   - Minimum value checkbox
   - Maximum value checkbox

   Note: The Operator menu, the configuration functions, and Operator text field are colored yellow, which indicate that configuration is to be completed or invalid.

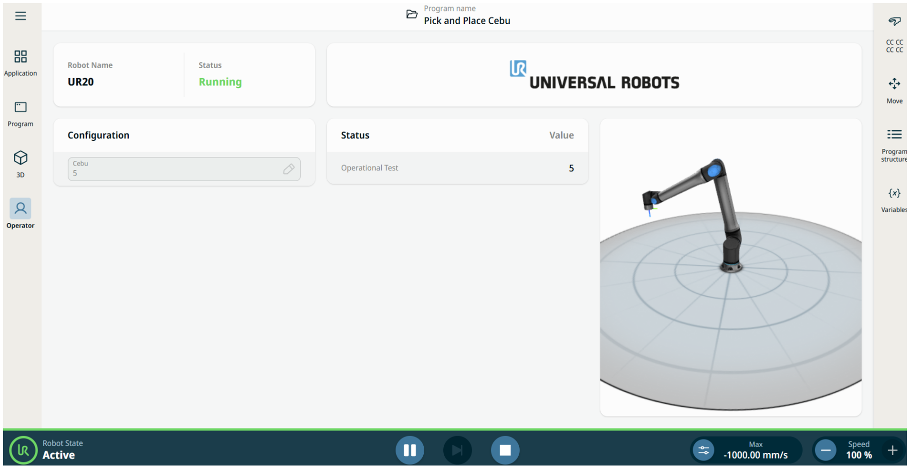


3. Tap the **Types** field, and four options are selectable:
    - Integer. For whole number.
    - Float. For decimal point value.
    - String. For text or a sequence of characters.
    - Boolean. Either true or false.
    - Dropdown

    When integer and float types are selected, a minimum and maximum value are specified.
4. Tap the **Initial Value** field to set the value. Note: This field does not appear when Boolean type is selected.
5. When the dropdown type is selected, a module/function pair can be picked.
6. Tap the **Operator Text** field, enter the name, and **Confirm**.

Adding a Status

1. On the left panel, tap **Status** under the **Sections** tree.
2. Tap **Add Status** on the center panel. A **Setup** right panel is inserted, similar to adding configuration.
3. Follow the iterative process numbers 1-3 in configuring the operator screen.
4. Go to the Operator Screen menu to see the configured operator screen.



22. Services (Interfaces)

Description

Services lists the standard services running on the robot. You can enable or disable each service.

> **NOTICE**
> All services are disabled as a security feature and are password protected. When you start or configure your robot, you have to unlock and enable relevant services.

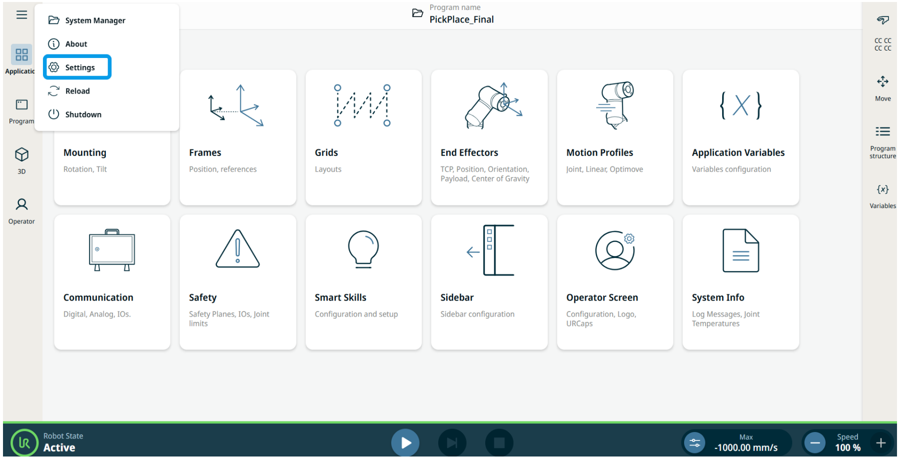
An enabled service remains available even if the ports associated with that service are blocked. So, simply blocking a port is not enough to prevent access to the listed services if they are enabled.

Enabling Services

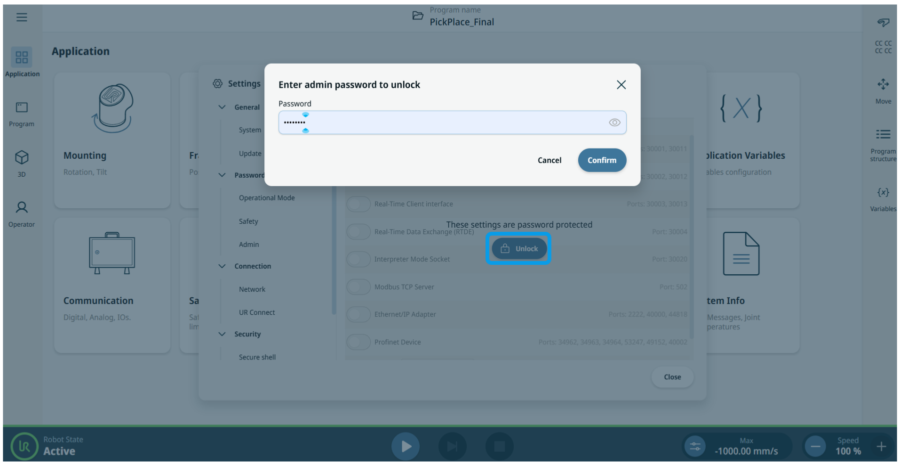
You have to enable the relevant service for the function that you are using.

To enable Services

1. In the main navigation, tap the **hamburger icon**.
2. Tap **Settings**.



3. Under **Security**, select **Services**.
4. Tap the **Unlock icon**, enter the admin password, and **Confirm**.



5. In the list, select an option and tap the slider button to enable (button turns blue) and revert to disable. Then **Lock and Close**.

22.1. Ethernet/IP Adapter

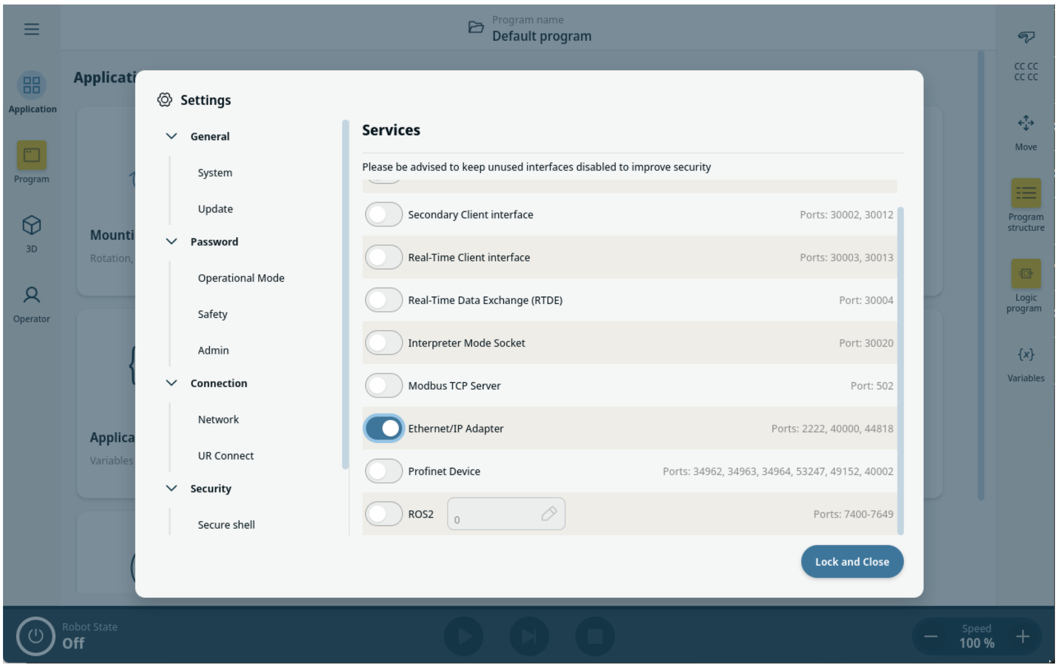
Description

EtherNet/IP is a network protocol that enables the connection of the robot to an industrial EtherNet/IP scanner device. If the connection is enabled, you can select the action that occurs when a program loses EtherNet/IP scanner device connection.

Enable Ethernet/IP

This is how to enable Ethernet/IP function in PolyScope X.

1. In the top left of the screen, tap the Hamburger menu.
2. Tap Settings.
3. In the menu on the left, under Security, tap Services.
4. Type the admin password.
5. Tap the Ethernet/IP Adapter button to switch toggle on.



Using Ethernet/IP

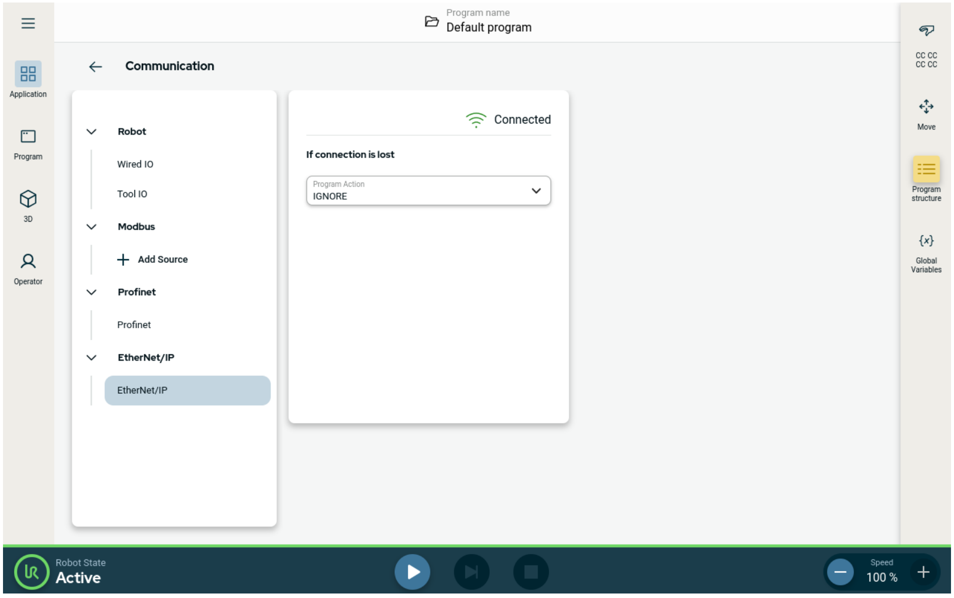
This is how you find the Ethernet/IP functions in PolyScope X:

In the PolyScope X left header.

1. Tap the Application icon.
2. Under Communication, select Ethernet/IP from the left menu.

Select the relevant action from the list:

Ignore	PolyScope X ignores the loss of EtherNet/IP connection, and the main program continues to run.
Pause	PolyScope X pauses the main program. The program resumes from where it stopped.
Stop	PolyScope X stops the main program.



In the upper right corner of this screen, you can see the Ethernet/IP status.

Connected	The robot is connected to the Ethernet/IP Scanner.
No Scanner	Ethernet/IP Device is running, but no Scanner is connected to the robot via Ethernet/IP.
Disabled	Ethernet/IP Device is not enabled.

22.2. PROFINET Device

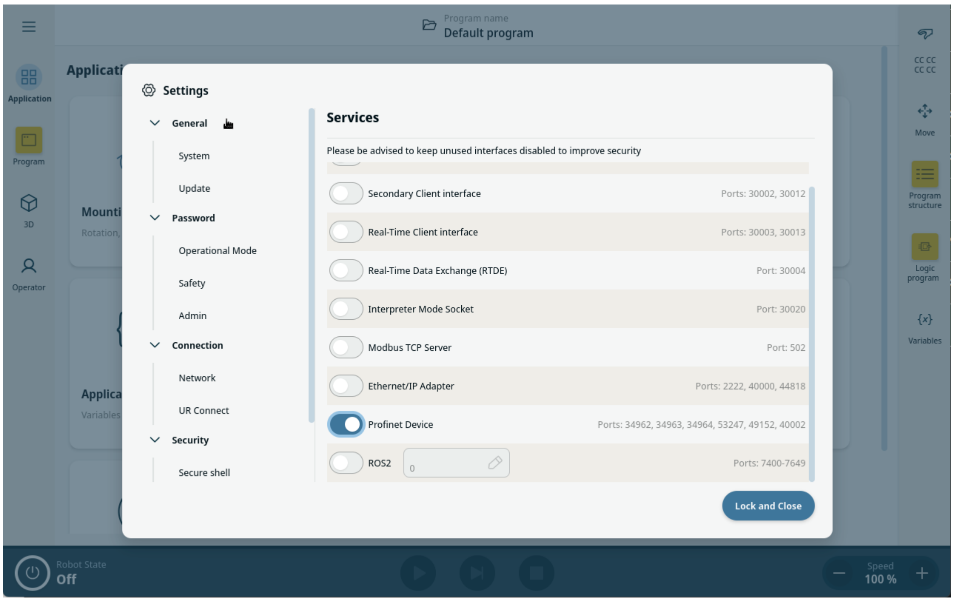
Description

The PROFINET network protocol enables or disables the connection of the robot to an industrial PROFINET IO-Controller. If the connection is enabled, you can select the action that occurs when a program loses PROFINET IO-Controller connection.

Enable PROFINET

This is how you enable PROFINET function in PolyScope X.

1. In the top left of the screen, tap the Hamburger menu and then tap Settings.
2. In the menu on the left, under Security, tap Services.
3. Type the admin password.
4. Tap the PROFINET button to switch PROFINET on.



Using PROFINET

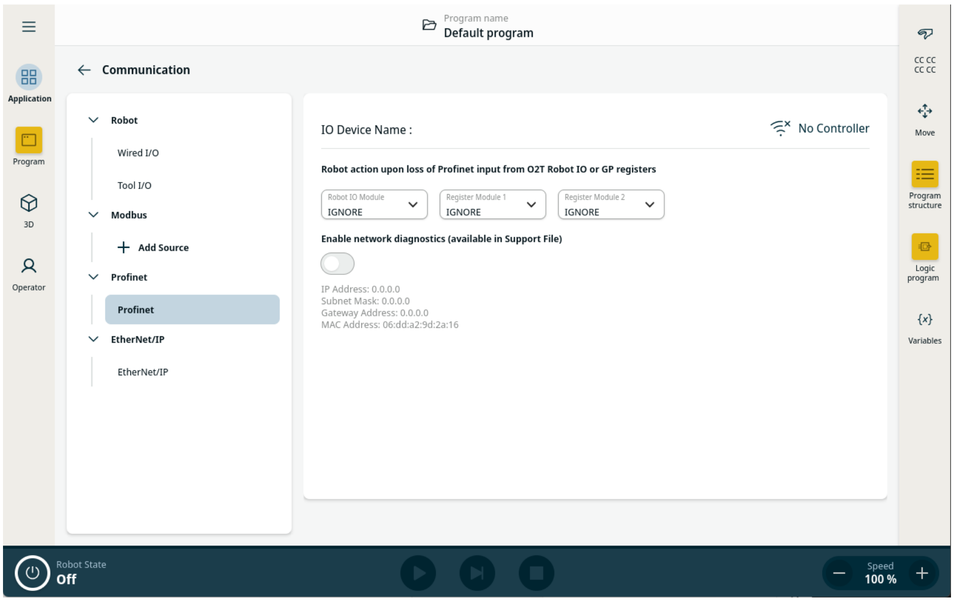
Find the PROFINET functions in PolyScope X:

In the PolyScope X Main Navigation.

1. Tap the Application icon.
2. Under Communication, select PROFINET from the left menu.

Select the relevant action from the list:

Ignore	PolyScope X ignores the loss of PROFINET connection, and the main program continues to run.
Pause	PolyScope X pauses the main program. The program resumes from where it stopped.
Stop	PolyScope X stops the main program.



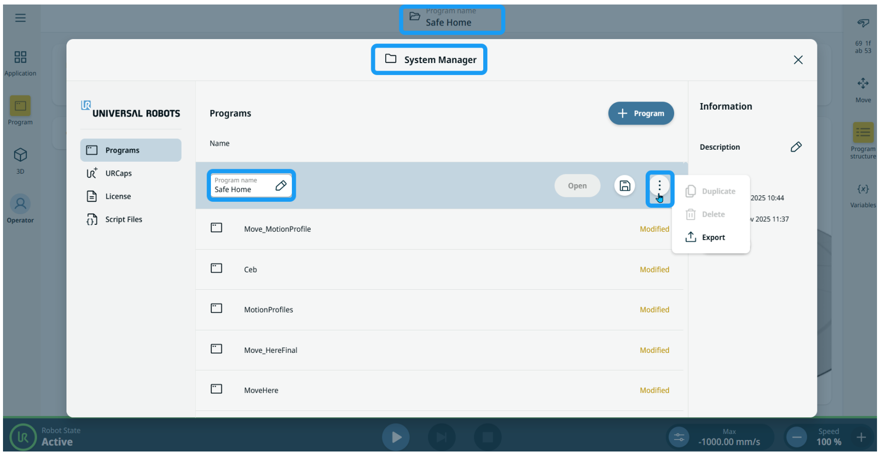
Diagnostics

Polyscope X has an option to record network traffic between Robot and PROFINET I/O-Controller. This can be used for diagnostics in case of connectivity issues.

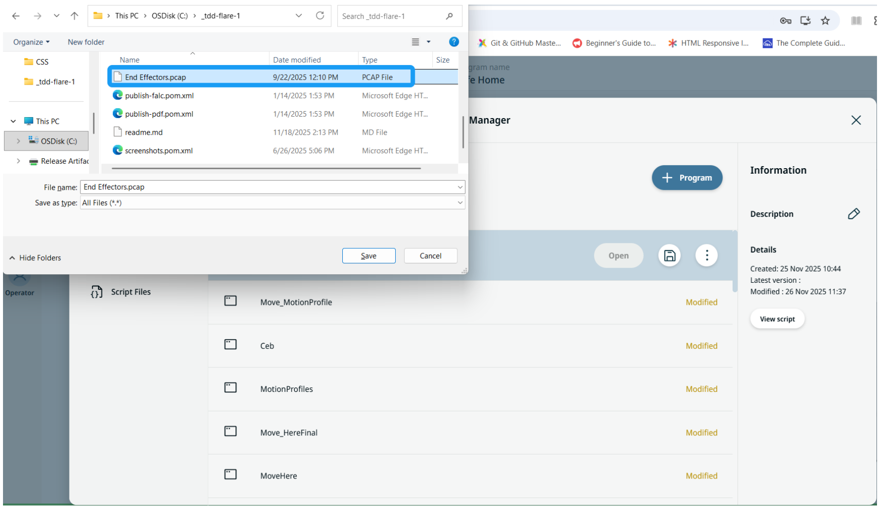
- To enable this option, tap the button "Enable network diagnostics" to toggle on.

Communication data will be saved in a .pcap file.
The file is saved in Support File in System Manager. Up to 50 MB of data can be recorded in the diagnostics.

1. Go to the **Header**, and the **System Manager** appears.
2. Choose the program you are working on.
3. Tap the three-vertical-dots icon (kebab icon) and choose **Export**.



4. Choose the communication data in pcap file and Save.



5. A pop-up notification appears in the main screen, stating you have successfully exported the file in the program.

Ethernet port status

When PROFINET Device is enabled, a new virtual ethernet port will be created.
Virtual ethernet port configuration shows information about currently configured IP address, subnet mask, gateway and MAC address.
Note that this virtual port is different from port configured in Robot network settings.

New PROFINET version

New GSDML file: GSDML-V2.45-UR-PROFIsafe-20251201.xml

NOTICE

Backwards compatibility:

- PLC programs made for previous software versions will run with no changes.
- Internal configuration file for PROFINET is copied, and the copy is migrated to the new PROFINET version.
- New PLC programs should use the recently released and proper GSDML file.

WARNING

It is not possible to downgrade to previous version. All changes made after the upgrade will be lost. Back up all data before upgrading.

Software Name: PolyScope X
Software Version: 10.12.1
Document Version: 10.17.188