UNIVERSAL ROBOTS

Software Handbook

PolyScope X

Original instructions (en)

SW 10.10

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.

3. 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.

3.1. 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.

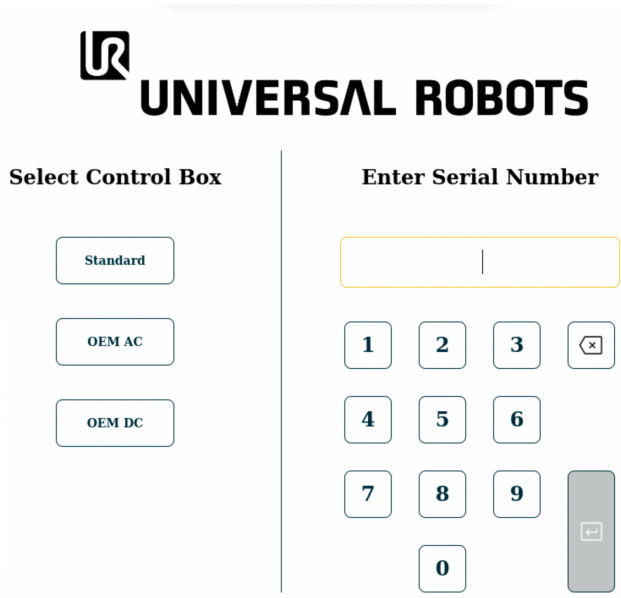
3.2. Inserting the Serial Number

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.

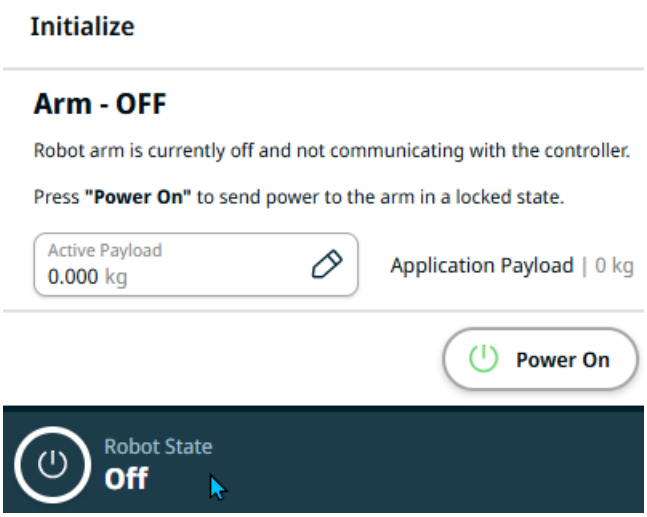


3.3. Starting the Robot Arm

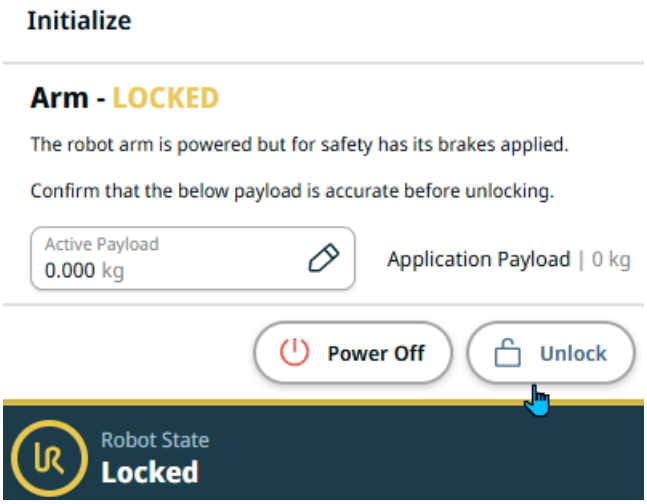
To start the robot

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**.

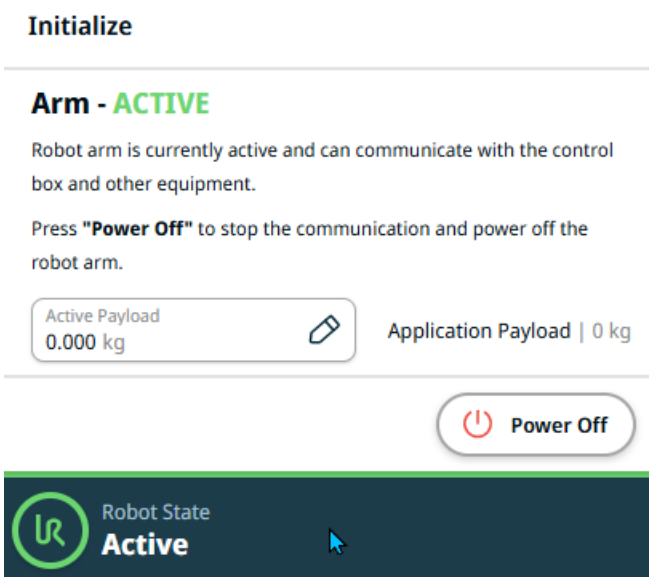


3. Tap **Unlock** to release the brakes.



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.

3.4. 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.

3.5. 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.

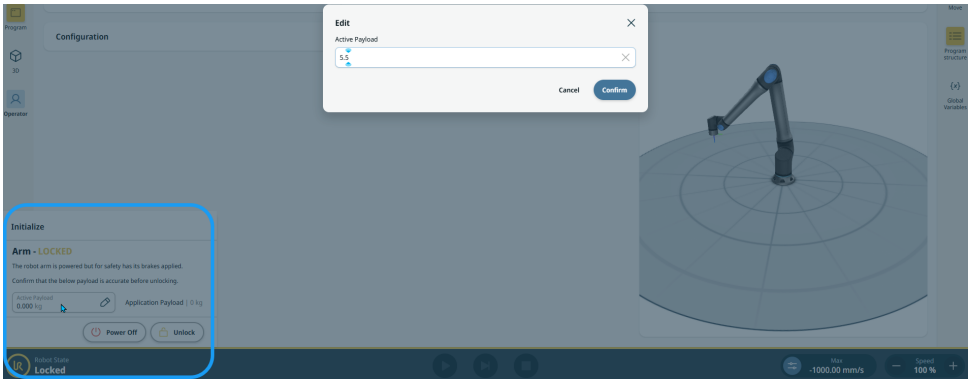
3.6. Safely Setting the Active Payload

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.

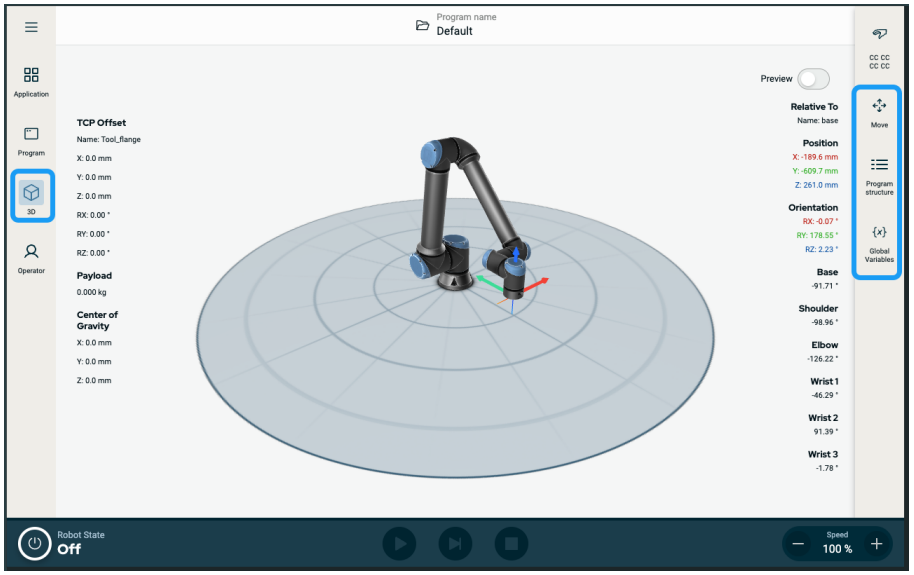
4. PolyScope X Overview

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.

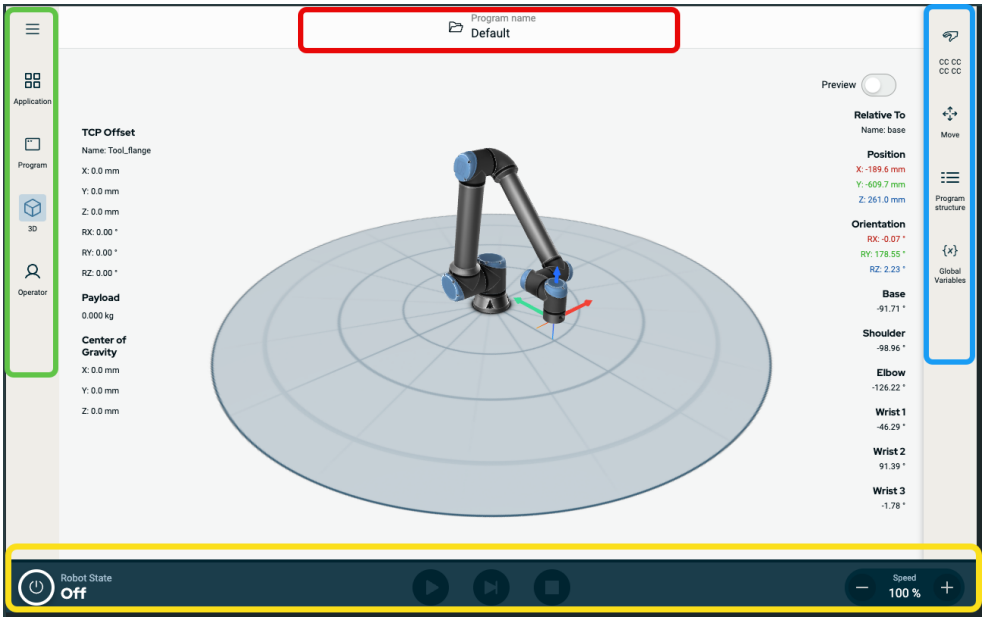
To view Main Screen

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.

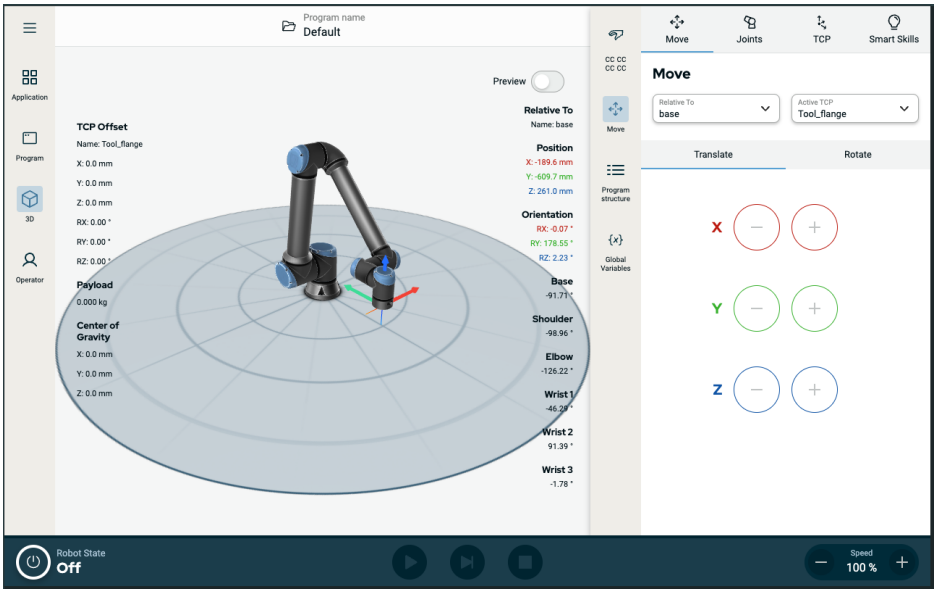


Figure 1.1: *Main screen and multitask screen*

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.

4.1. Icons

4.1.1. Header Icon

Icon	Title	Description
	Program name	Gives access to System Manager. Enables you to create, modify, add program, and URCaps files.

4.1.2. Left Toolbar Icons

Icon	Title	Description
	More	Access to information of robot version, serial number, and settings.
	Application	Configures and sets up the robot arm settings and safety, including end effectors and communication.
	Program	Access to basic and advanced 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.

4.1.3. Icons Inside the More/Hamburger Icon

Icon	Title	Description
	System Manager	Gives access to System Manager. Enables you to create, modify, add program, 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.

4.1.4. Right Toolbar Icons

Icon	Title	Description
	Safety Checksum	Access to the active safety checksum and 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 an orderly structure of created program(s). Access to add modules.
	Global Variables	Access to created program names and values.

4.1.5. Footer Icons/Buttons

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 slider	Manages the robot state. When RED, press it to make the robot operational.
	High Speed Manual	The High Speed Manual slider is only accessible 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.

4.1.6. 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.

4.2. 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.

4.3. Header

Description

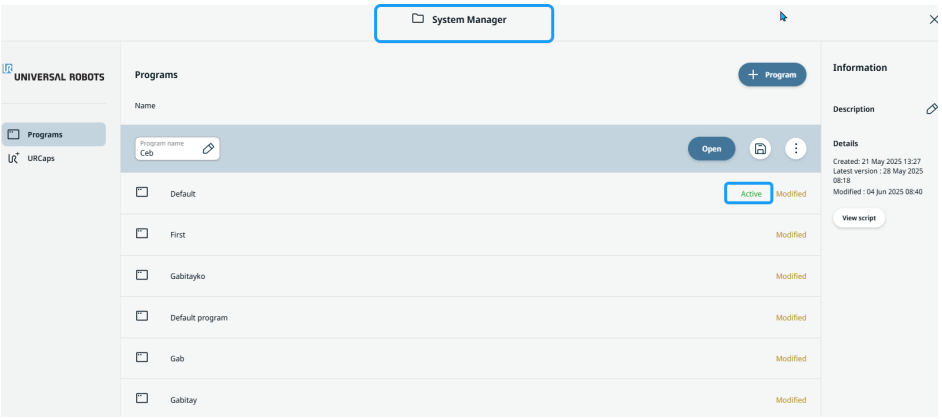
The header solely contains the **Program name** menu.

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
- Export a program
- Load a program

- Install URCaps
- Write a program description
- See program details
- View script



4.4. Main Navigation

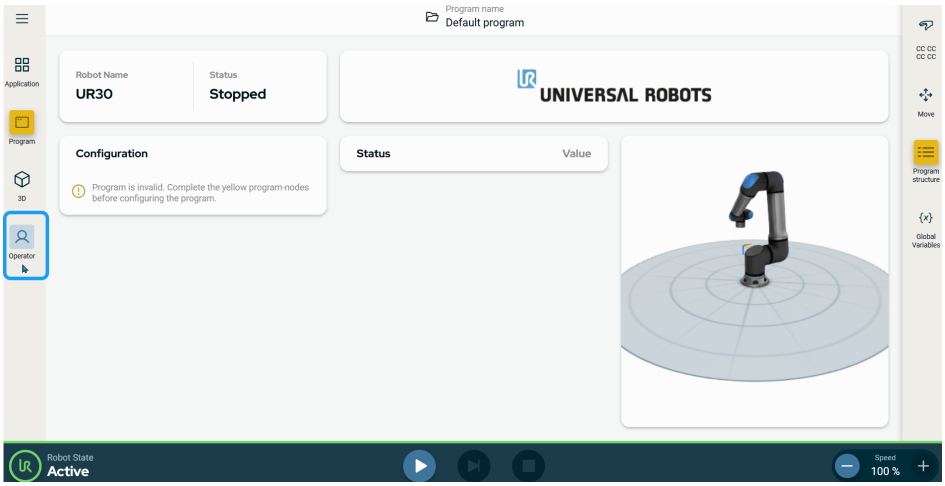
Description

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

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

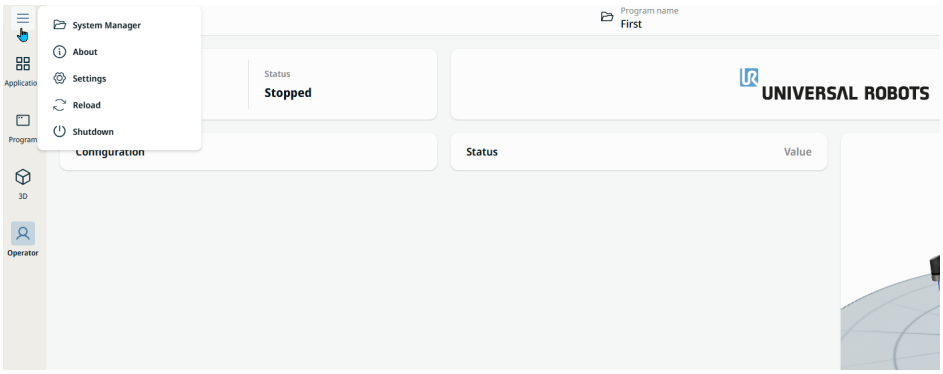
More about the Main Navigation

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

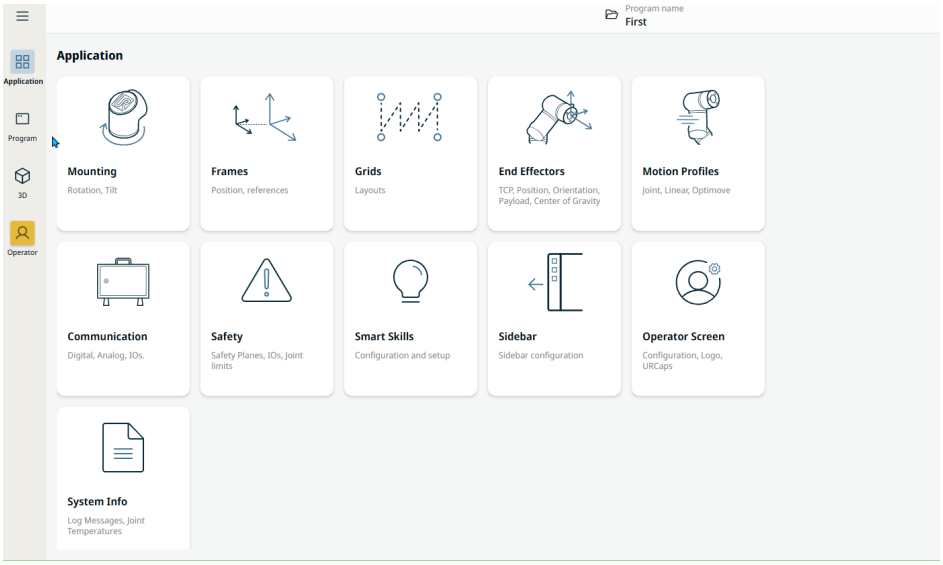


When you tap an icon, it is highlighted in blue, which indicates active in the main screen. The hamburger icon is an exception to this.

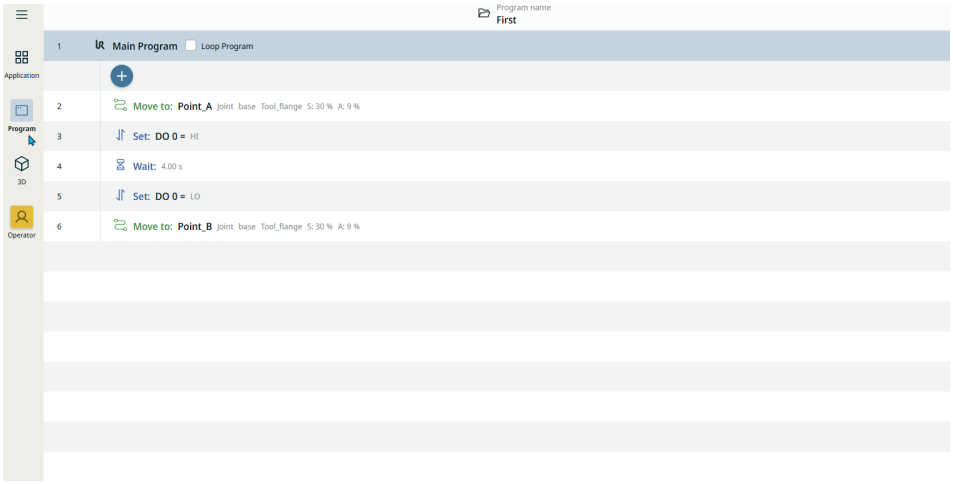
Tap hamburger icon



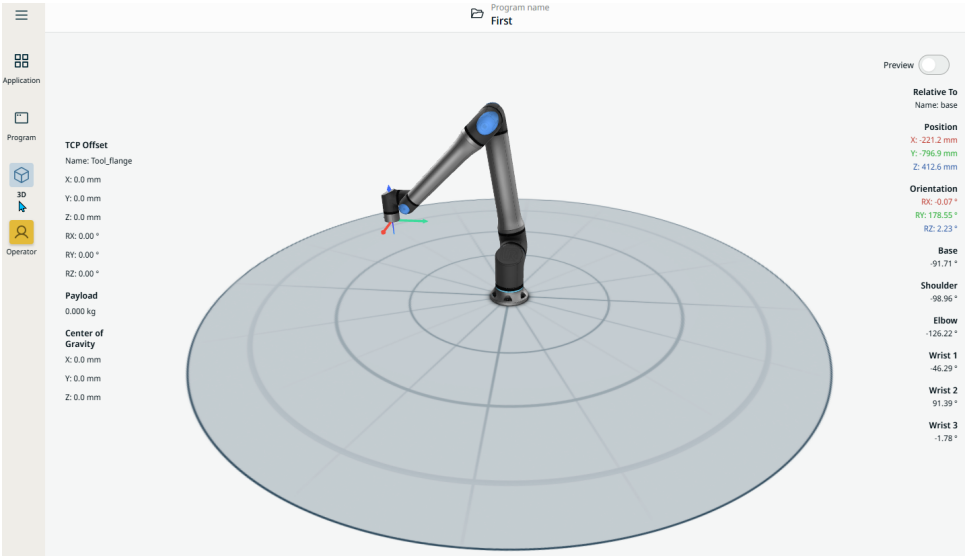
Tap Application



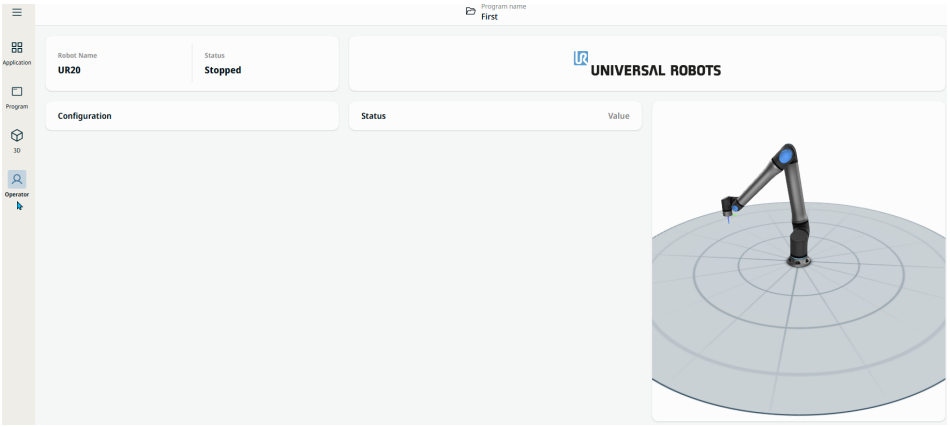
Tap Program



Tap 3D



Tap Operator

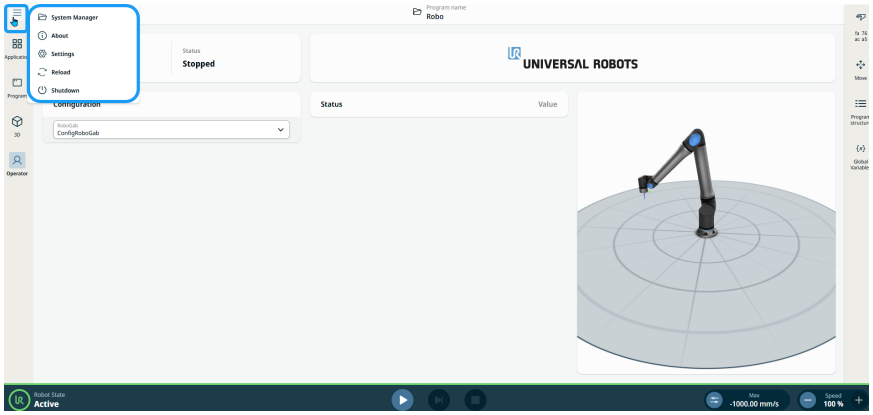


4.4.1. Functions Inside the Hamburger Icon

Description

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

- System Manager
- About
- Settings
- Reload
- Shutdown



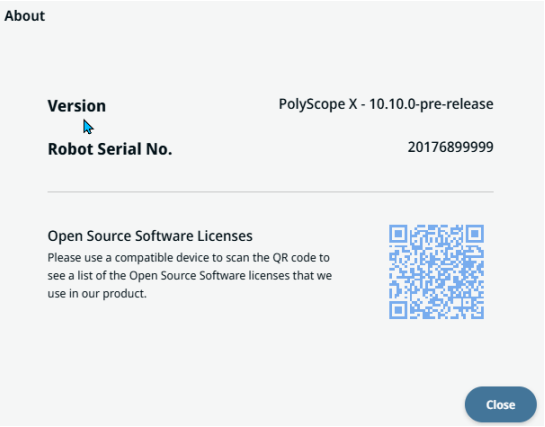
System Manager

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

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

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.



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.

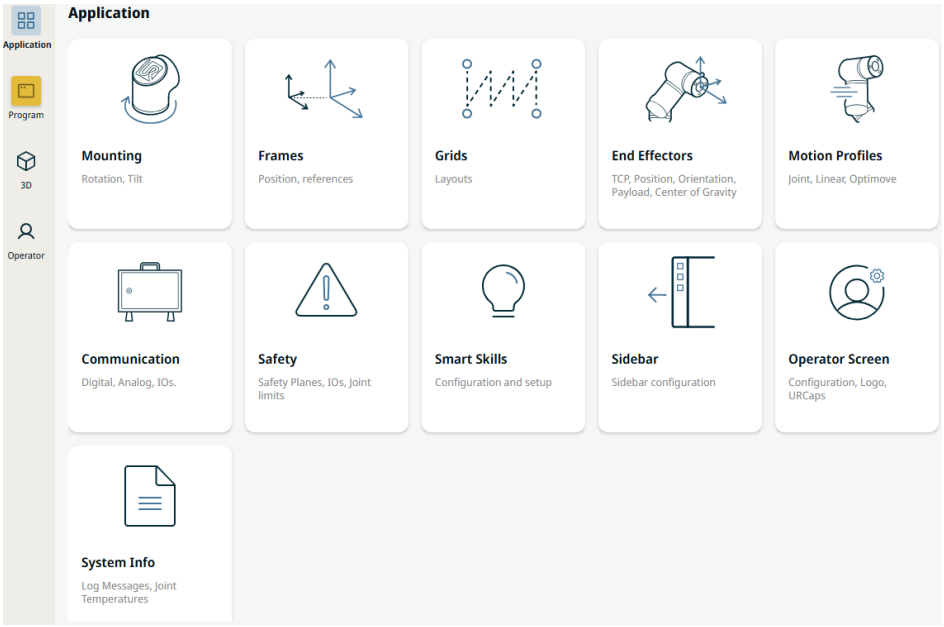
Shutdown The **Shutdown** function requires no further action to perform in another field or tab. When you tap Shutdown, you are given the choice to restart or power off. Program state will be saved before restarting or powering off the robot.

4.4.2. 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 eleven application submenus to use for robot movement, configuration, and information.

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

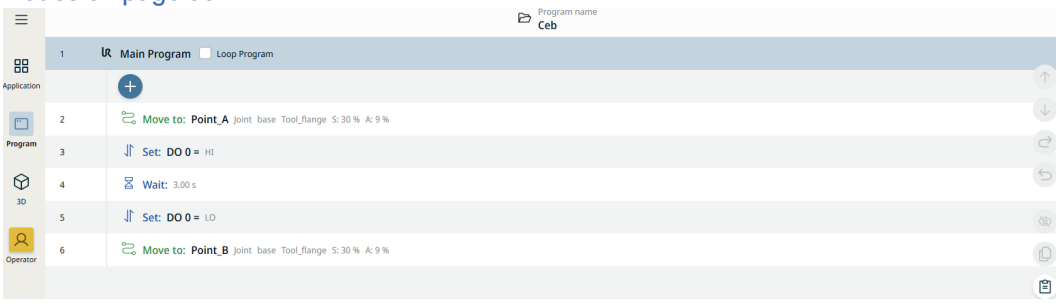
4.4.3. Program Menu

Description

The **Program** menu gives you access to the commands and clipboard toolboxes, which contains 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 13.3 Command Nodes on page 99.



4.4.4. 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**:

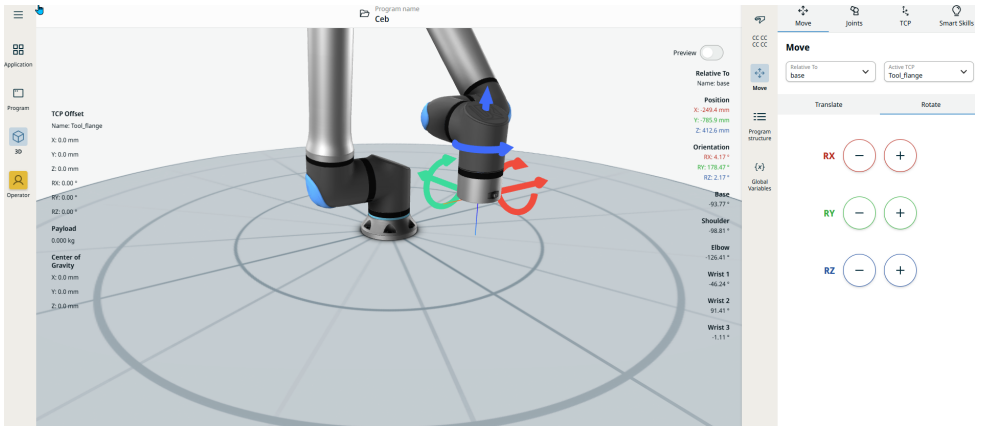
- **Press and rotate** clockwise or counterclockwise.

  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.



4.4.5. 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.

4.5. Sidebar

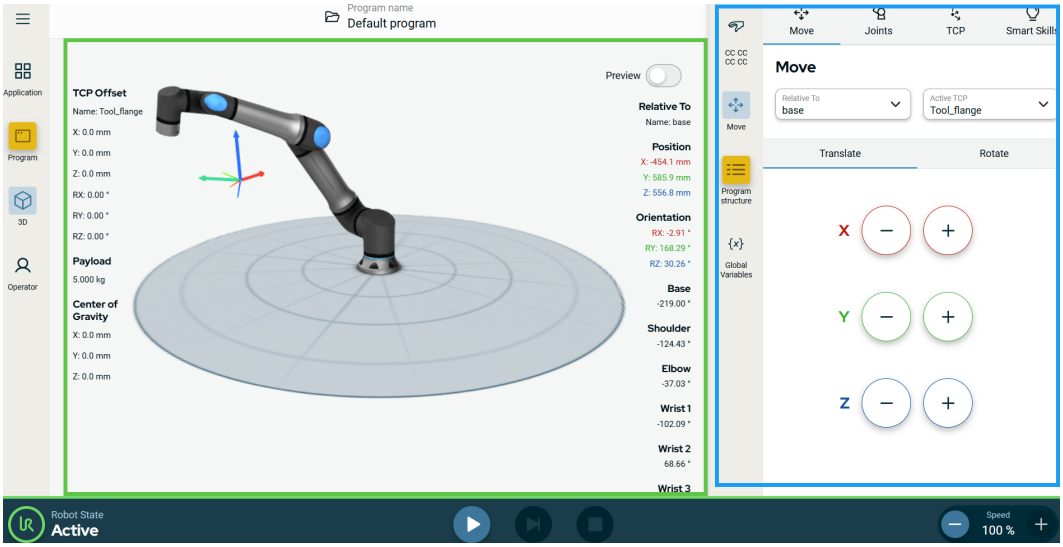
Description

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).



4.5.1. 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:

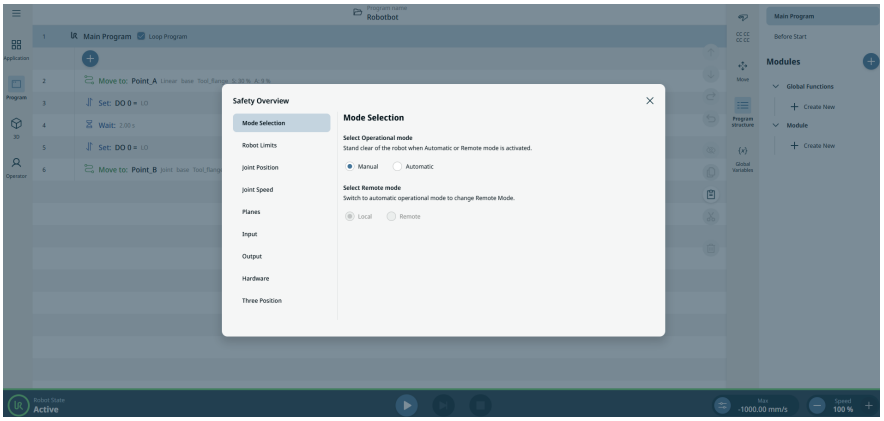
- Left side is smaller and contains the following safety item settings:

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



4.5.2. Move Menu

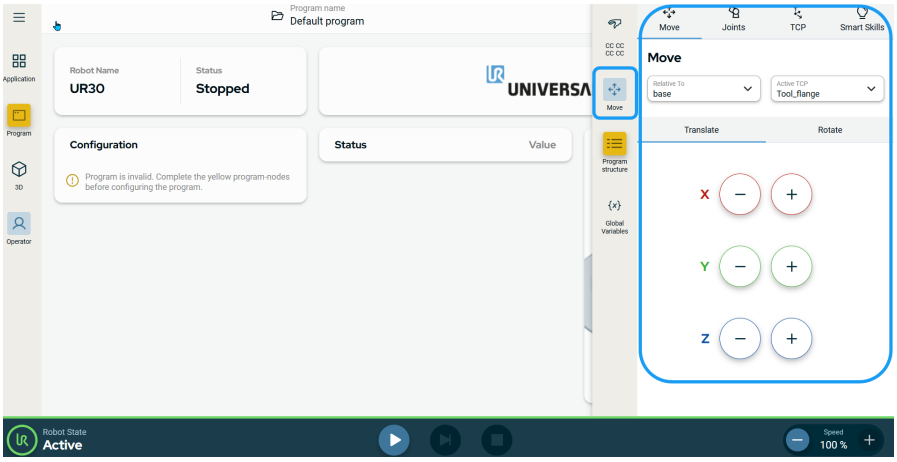
Description

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.

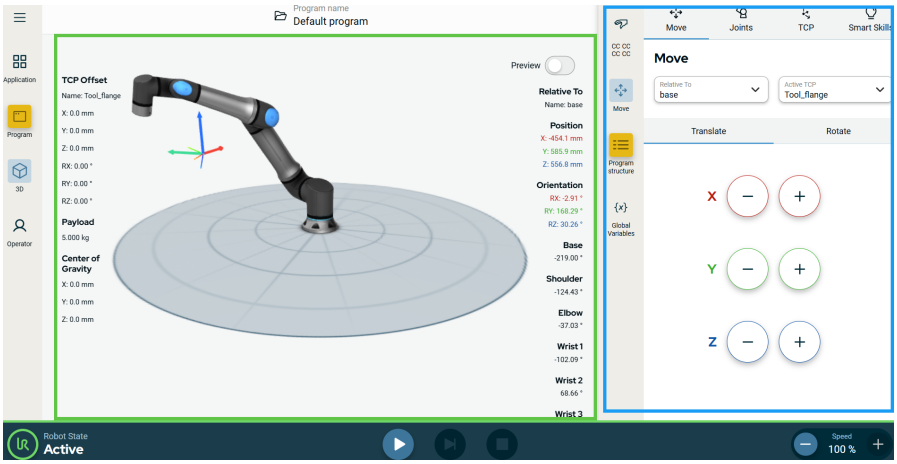
   The multitask screen expands to the center. You can see the fields and tabs related to the move menu.



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.

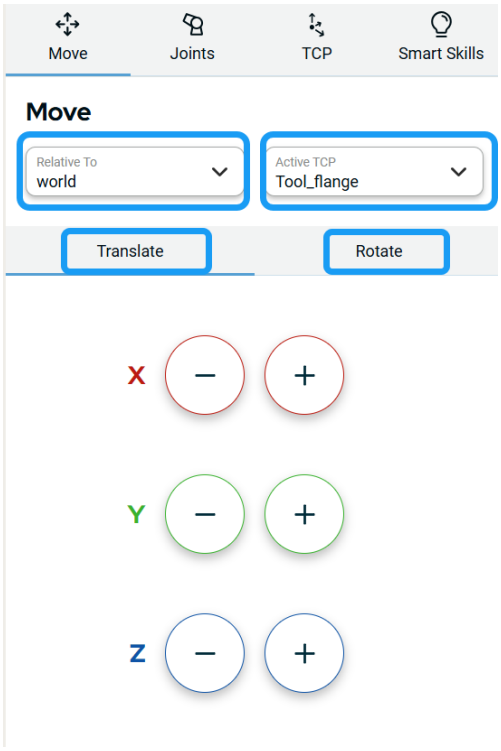
   In the main screen, you can see the corresponding movement(s) of the robot with what you input in the multitask screen.



Using the Move tool

The Move tool contains two fields and two tabs:

- Relative To
- Active TCP
- Translate
- Rotate



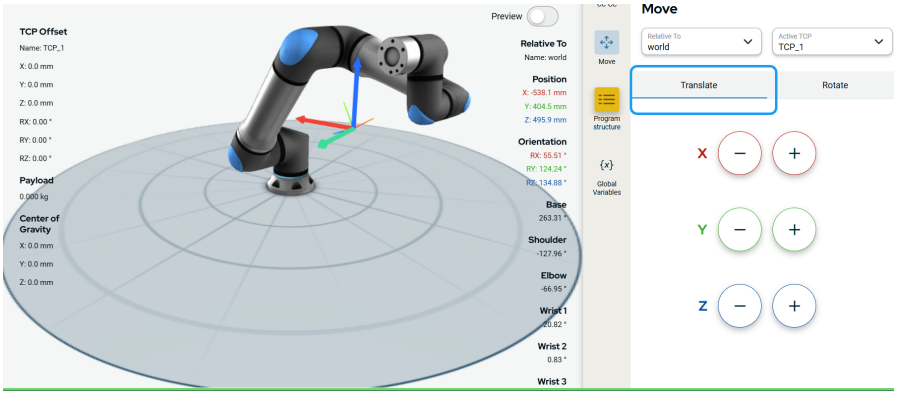
1. Tap the **Relative to** field.

   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.

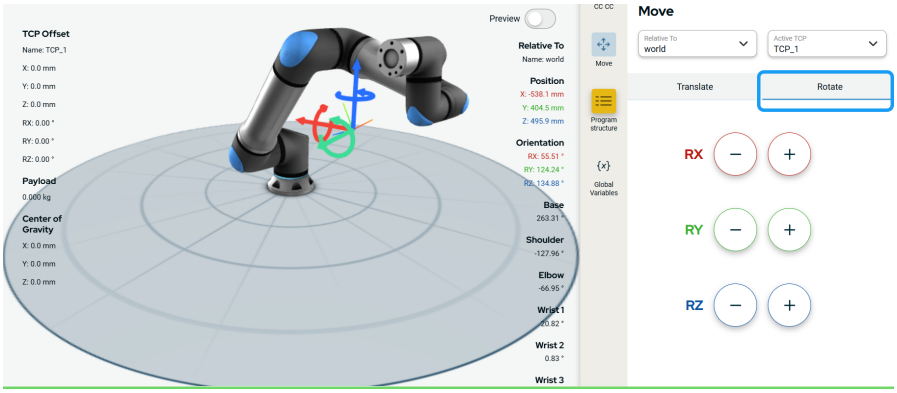
4. Tap the **Translate** tab.
5. Tap the **plus-minus** button of X, Y, Z coordinates to move the robot arm in an indicated direction.

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



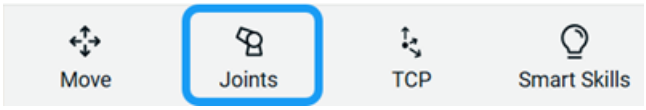
6. Tap the **Rotate** tab.
7. Tap the **plus-minus button** of RX, RY, RZ coordinates to change the orientation of the robot in the indicated direction.

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



Using the Joints tool

1. Tap **Joints**.

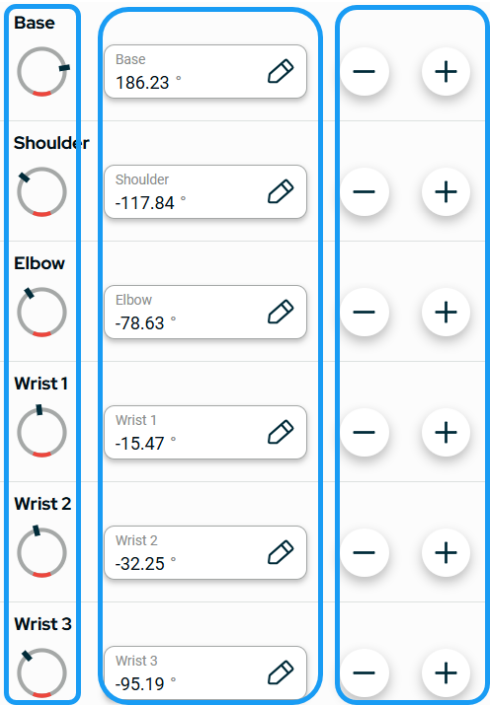


The Joints screen appears, which contains the six parts of the robot arm:

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

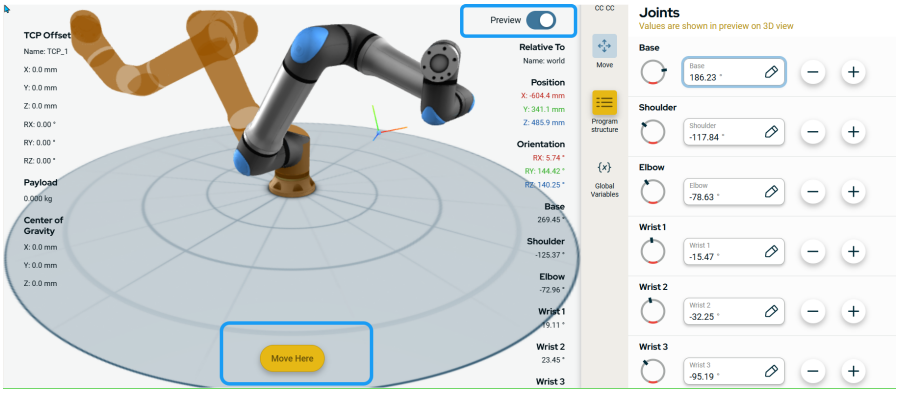
Each robot part gives you three adjustment tools so the position measurement be accurately achieved:

- 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.

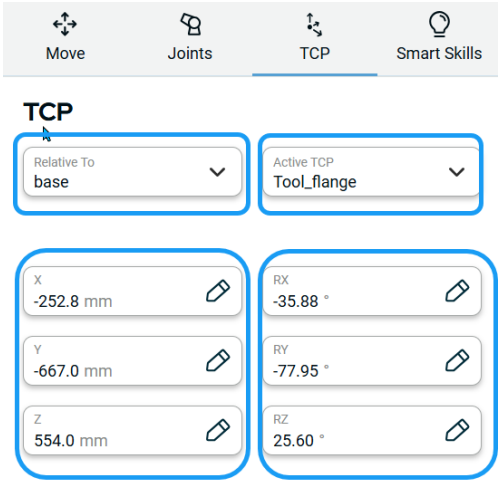
5. When you want to preview the whole robot arm movement, tap the brown **Move Here** button below the X, Y, Z coordinate space of the robot arm.
6. Tap the **plus-minus** button to adjust and reach your preferred measurement of the robot arm joints.

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



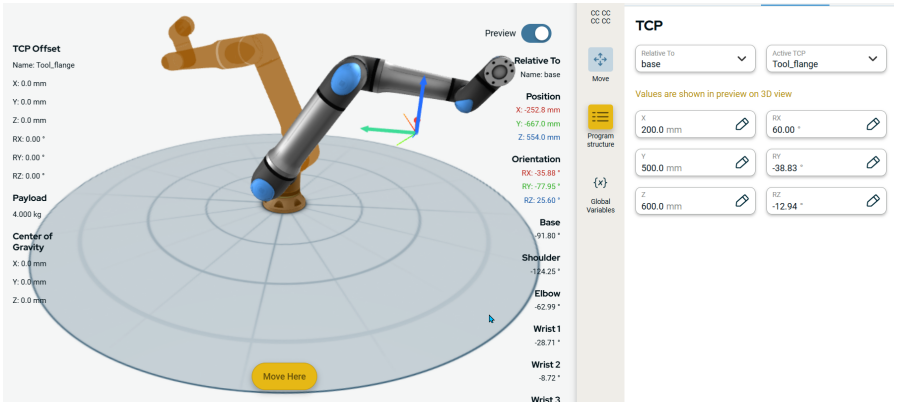
2. Tap the **Relative to** field.

   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.

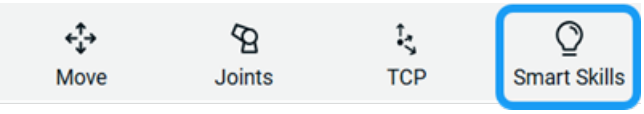
   A preview of the robot's position is shown in the robot arm with brown color.



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.

Using the Smart Skills tool

1. Tap **Smart Skills**.



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

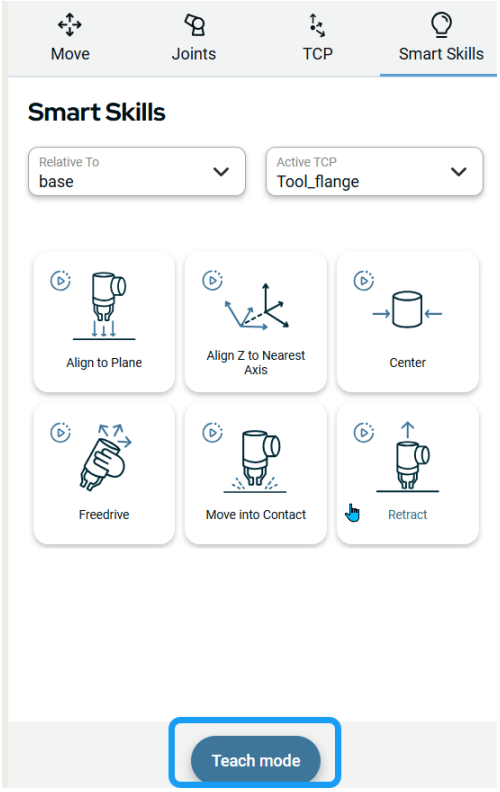
The two fields are:

- 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.

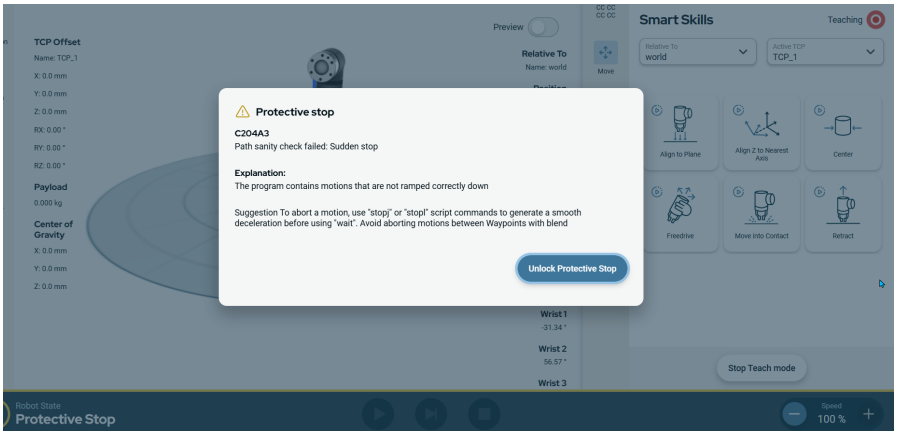
6. Tap the **Teach mode** button to program the robot by demonstration.

   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.



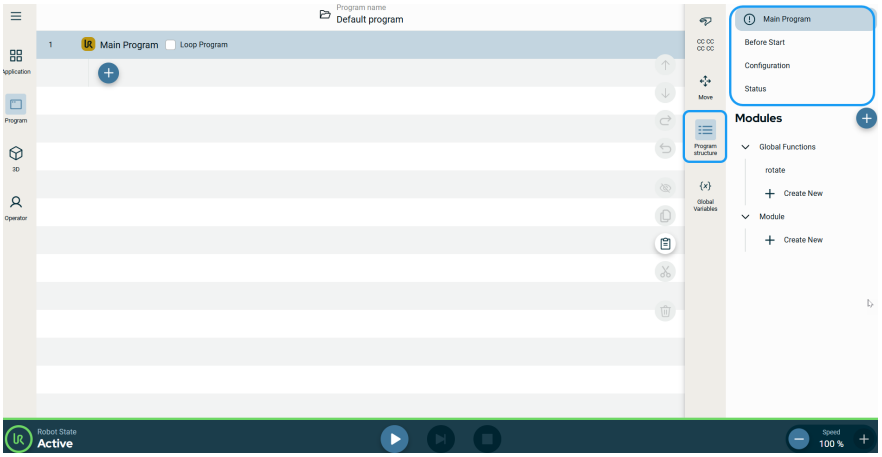
4.5.3. Program Structure

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**.

4.5.4. 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:

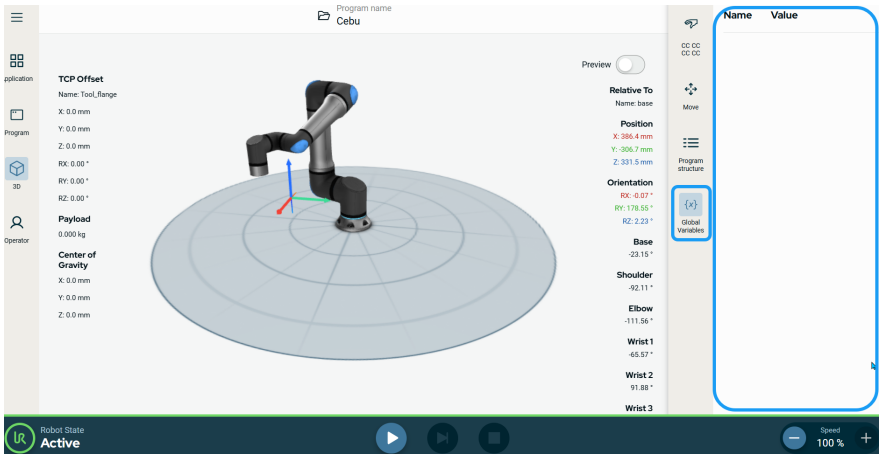
- Assignment nodes
- Operator Screen configuration
- Status
- Frames
- Waypoints

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

To access Global Variables

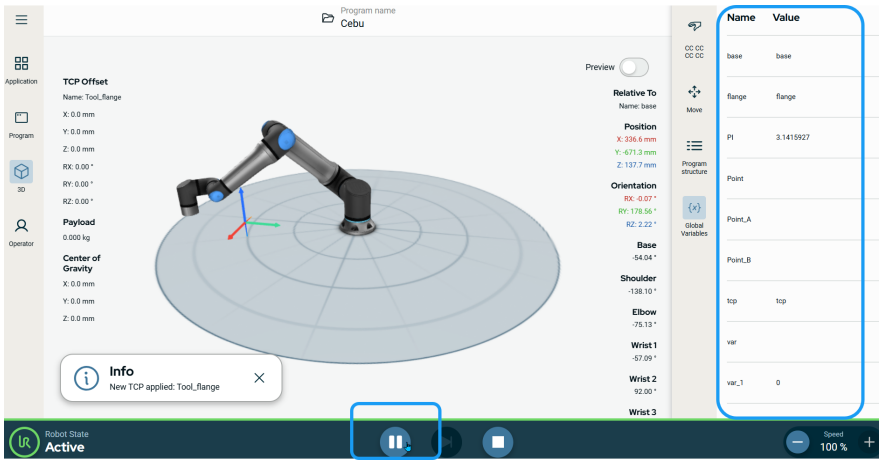
1. You can choose to view the main screen in **3D** view or **Operator** screen.
2. Tap **Global Variables** in the sidebar.

   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.



4.6. Footer

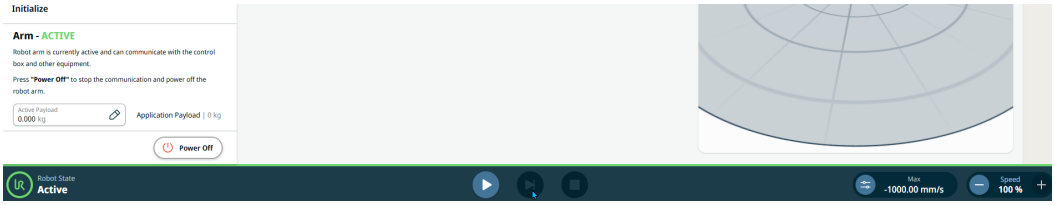
Description

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



5. Safety

Warning

WARNING

Before you configure your robot safety settings, your integrator must conduct a risk assessment to guarantee the safety of personnel and equipment around the robot. A risk assessment is an evaluation of all work procedures throughout the robot lifetime, conducted in order to apply correct safety configuration settings. (See Hardware Installation Manual)

Safety Checksum

The Safety Checksum icon displays your applied robot safety configuration. The checksum changes if and only if the safety configuration is changed.

Safety Configuration

NOTICE

Safety Settings are password protected.

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. Note: Once Safety settings are unlocked, all settings are now active.
5. Tap LOCK or navigate away from the Safety menu to lock all Safety item settings again.

Setting a Safety Password

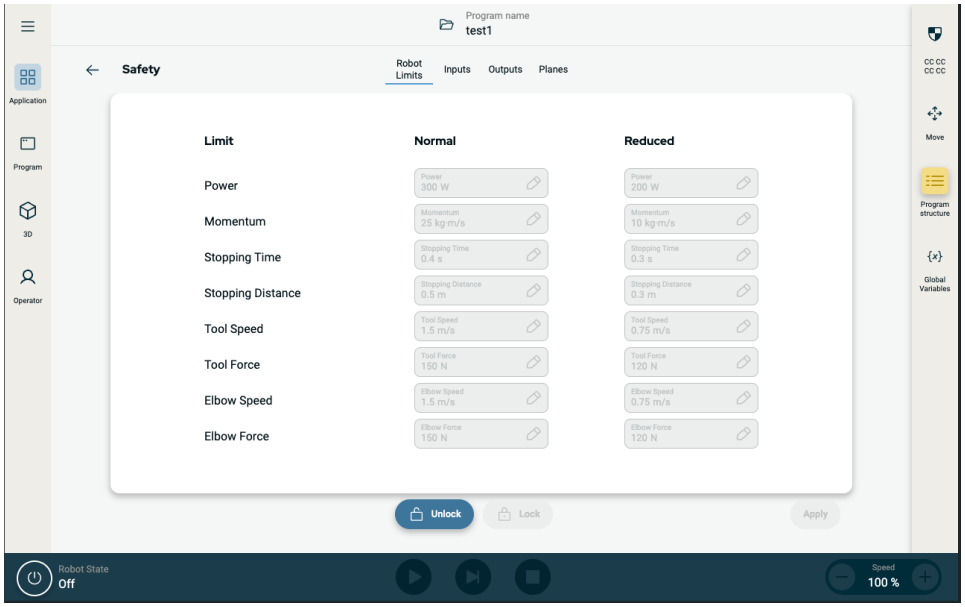
1. In your PolyScope X header left corner, tap the Hamburger menu and then tap Settings.
2. On the left of the screen, in the blue menu, tap Safety Password.
3. For Old Password, type the current Safety password.
4. For New Password, type a password.
5. For Repeat Password, type the same password and tap Change Password.
6. In the top right of the menu, press CLOSE to return to previous screen.

5.1. Safety Menu Settings

Safety First Here you can find descriptions of the safety settings in PolyScope X.

5.1.1. Robot Limits

Limits



Limit	Description
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.
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.
Tool Speed	Limits maximum robot tool speed.
Tool Force	Limits the maximum force exerted by the robot tool in clamping situations.
Elbow Speed	Limits maximum robot elbow speed.
Elbow Force	Limits maximum force that the elbow exerts on the environment.

Safety Mode

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.

NOTICE

The tool speed and force are limited at the tool flange and the center of the two user-defined tool positions

Under normal conditions, i.e. when no Robot stop is in effect, the safety system operates in a Safety Mode associated with a set of safety limits [1]:

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.

[1] Robot stop was previously known as "Protective stop" for Universal Robots.

5.1.2. Safety I/O Signals

Description

The I/O are divided between inputs and outputs and are paired up so that each function provides a Category 3 PLd capability.

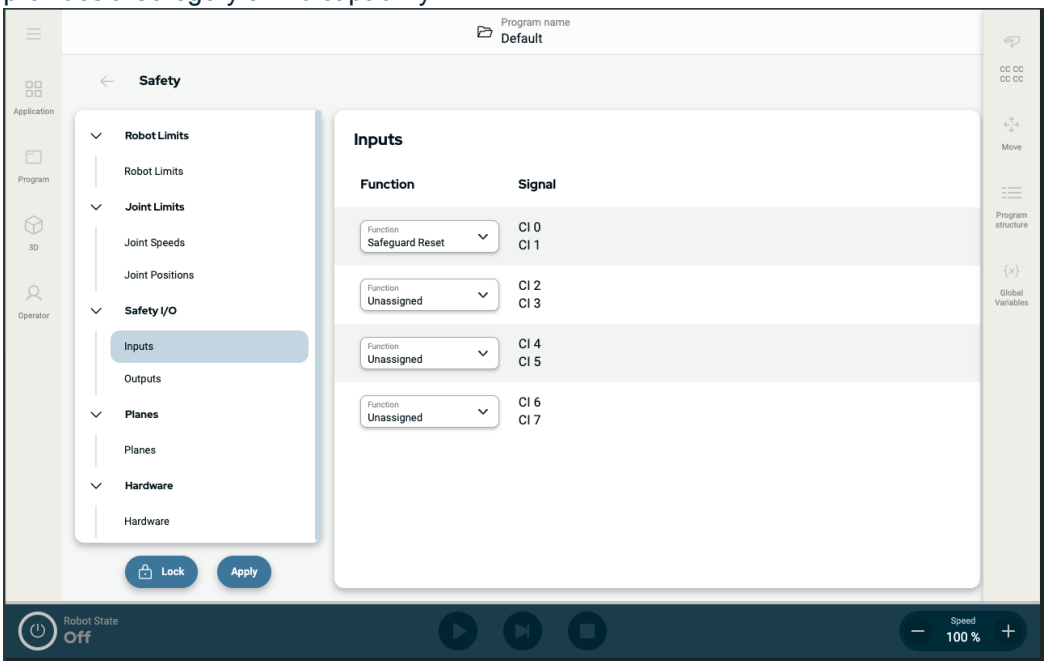
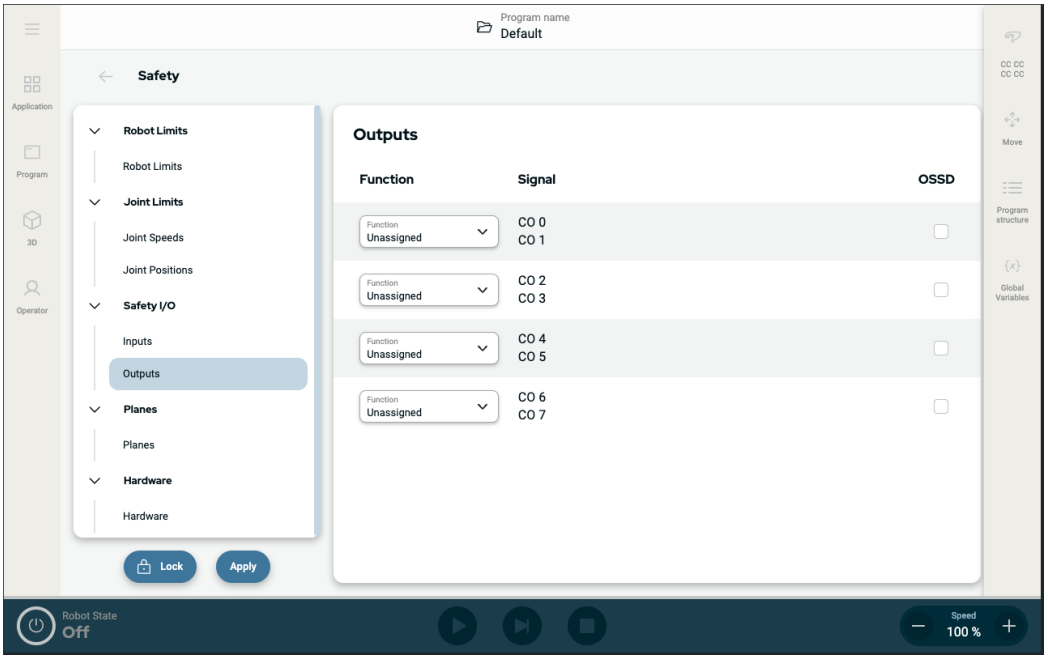


Figure 1.2: *PolyScope X screen displaying the Input signals.*



Input Signals

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.

Input Signals

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.

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.

Output Signals

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.
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.
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.

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.

[1]System Stop was previously known as "System Emergency Stop" for Universal Robots robots. PolyScope can display "System Emergency Stop".

5.1.3. Safety 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.

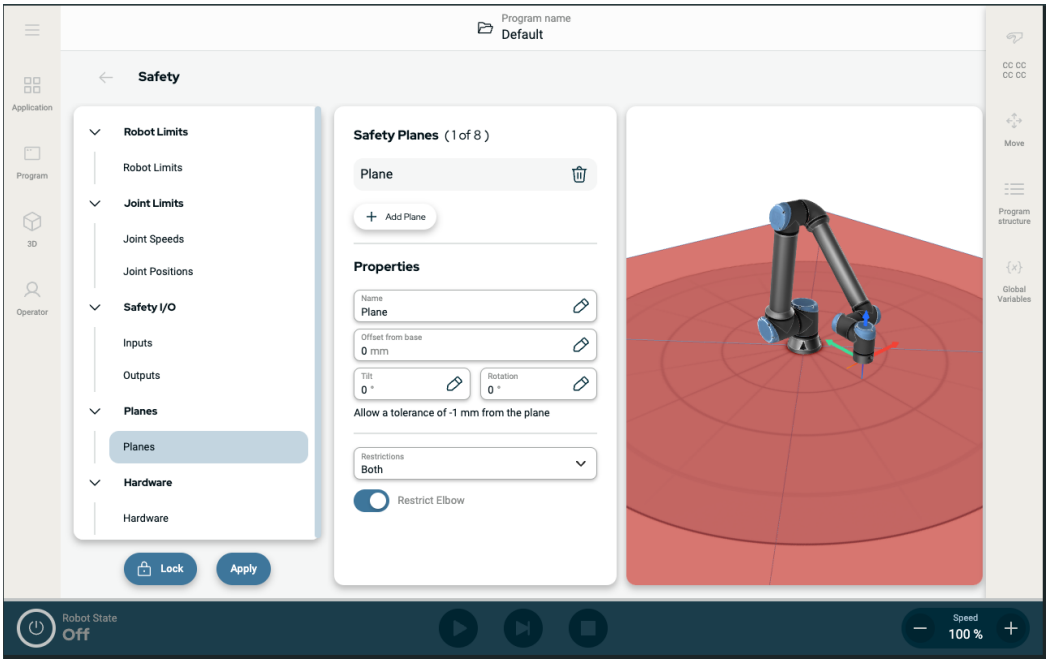


Figure 1.3: *PolyScope X screen displaying safety planes.*

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

You can prevent the robot elbow joint from passing through any of your defined planes.

Disable Restrict Elbow for elbow to pass through planes.

6. 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.

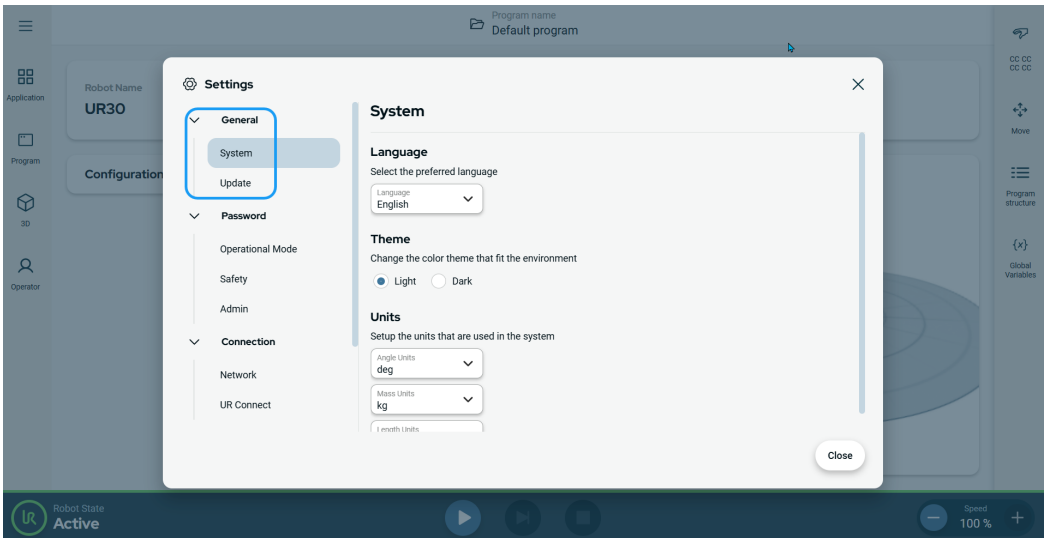
Security Settings

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

6.1. General Settings

Description

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



To open General setting

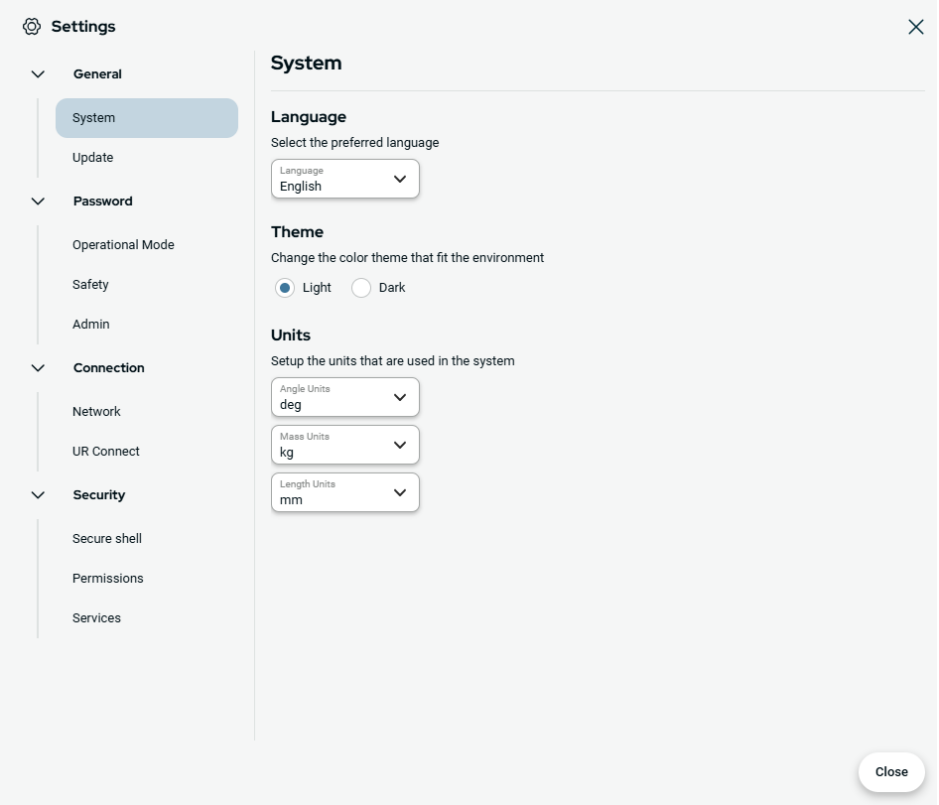
1. Tap the hamburger menu on the main navigation.
2. Tap **Settings** tab.

   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.

6.1.1. 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.

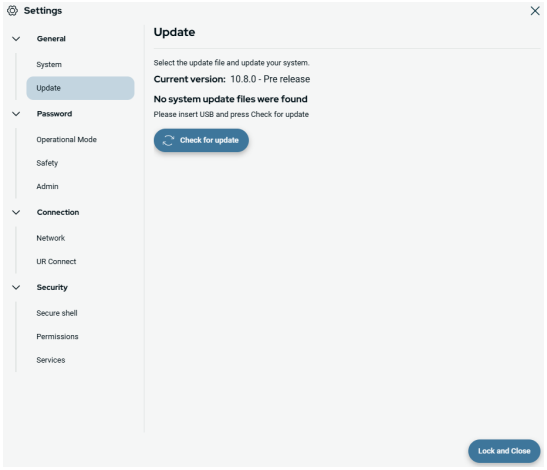
Length Units:
You can change between meter and millimeter.

6.1.2. Update

Description When you receive your robot, it is recommended that you update the software to the most recent released from Universal Robots.

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. Tab the button "Check for update".



6.2. 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.

6.2.1. 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

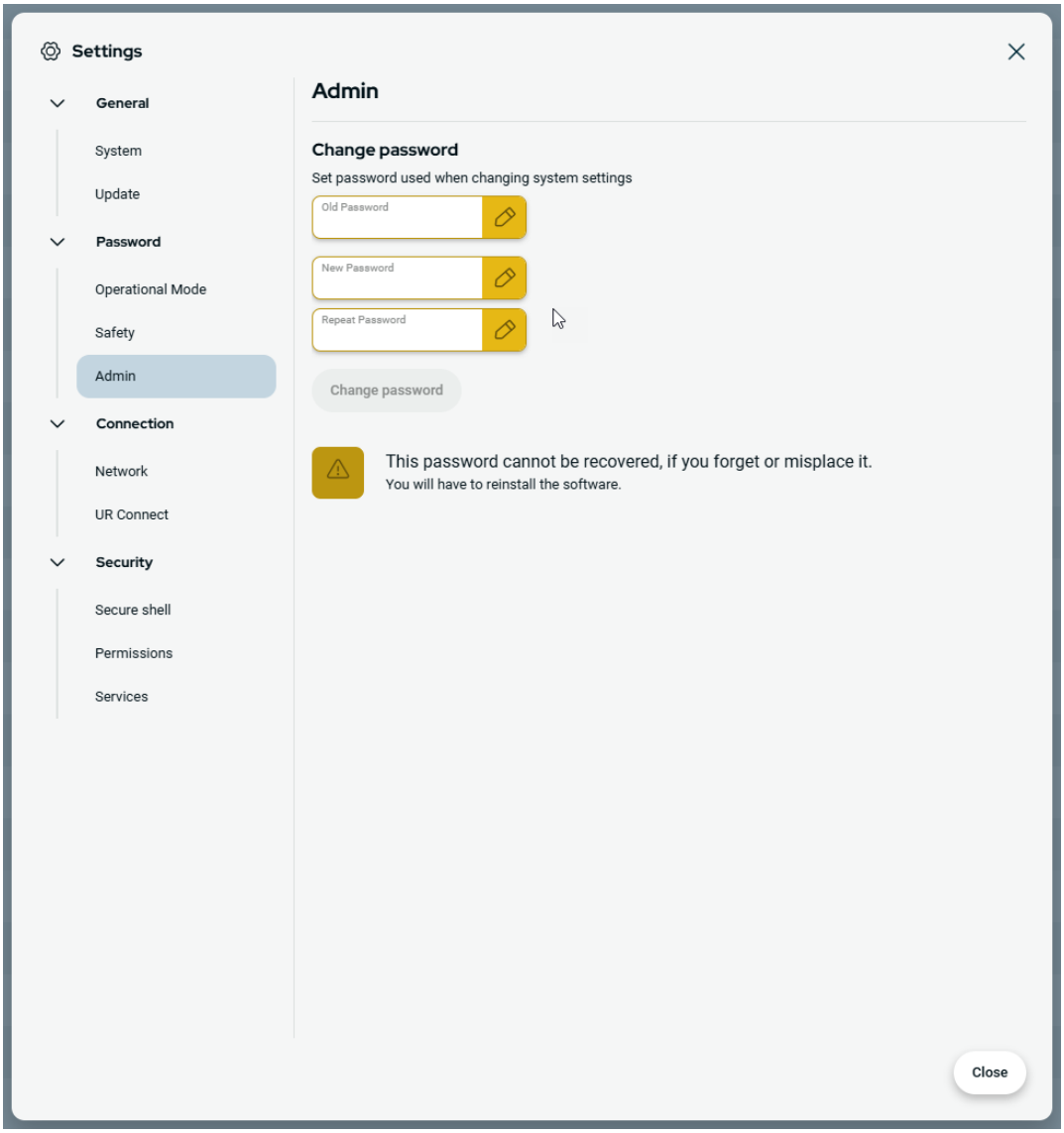
NOTICE

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.

6.2.2. Password – Operational Mode

Default Password

The default password for operational mode: operator

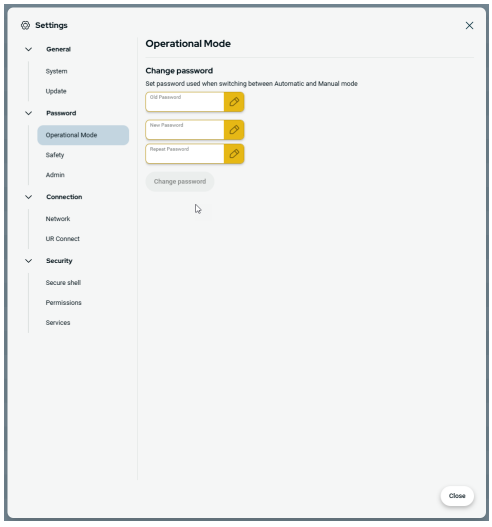
> **NOTICE**
> If you forget your password, it cannot be replaced or recovered.
> You will have to reinstall the software.

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.



6.2.3. Password – Safety

Default Password

The default password for safety: ursafe

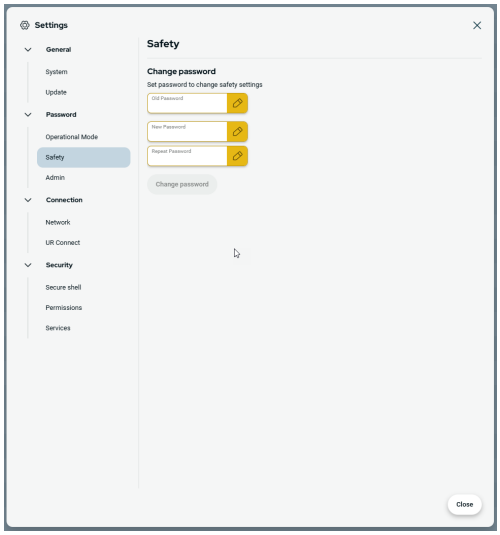
> **NOTICE**
> If you forget your password, it cannot be replaced or recovered.
> You will have to reinstall the software.

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.



6.3. 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

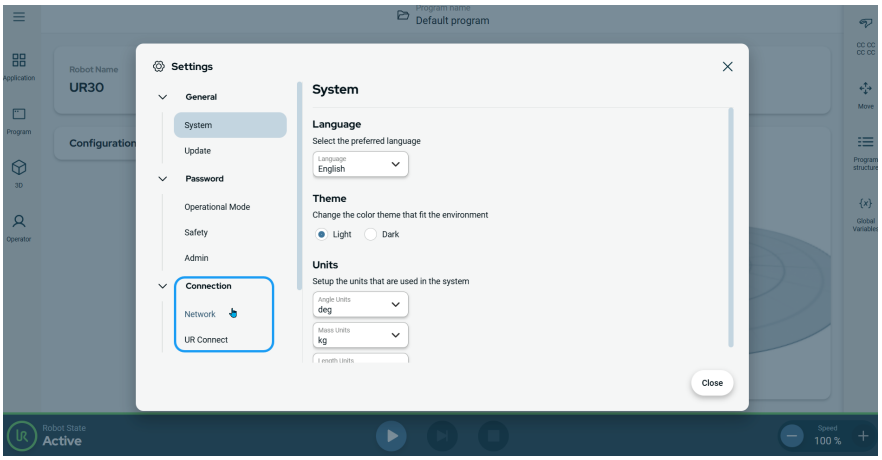
The main screen can be seen below.

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.



6.3.1. Network

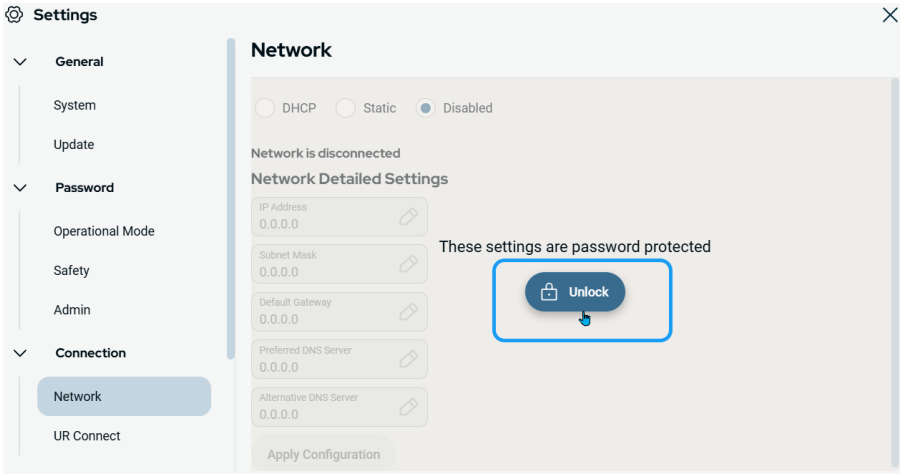
Description

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

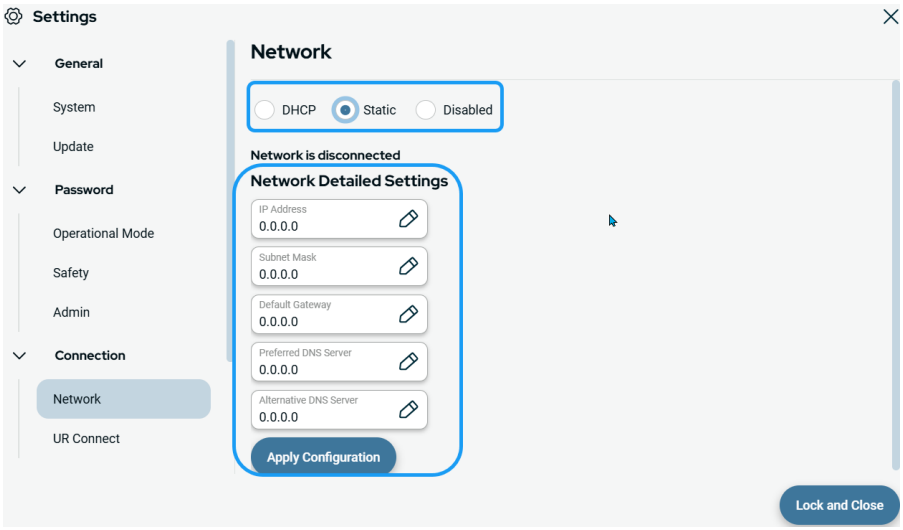
To connect via Network

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.

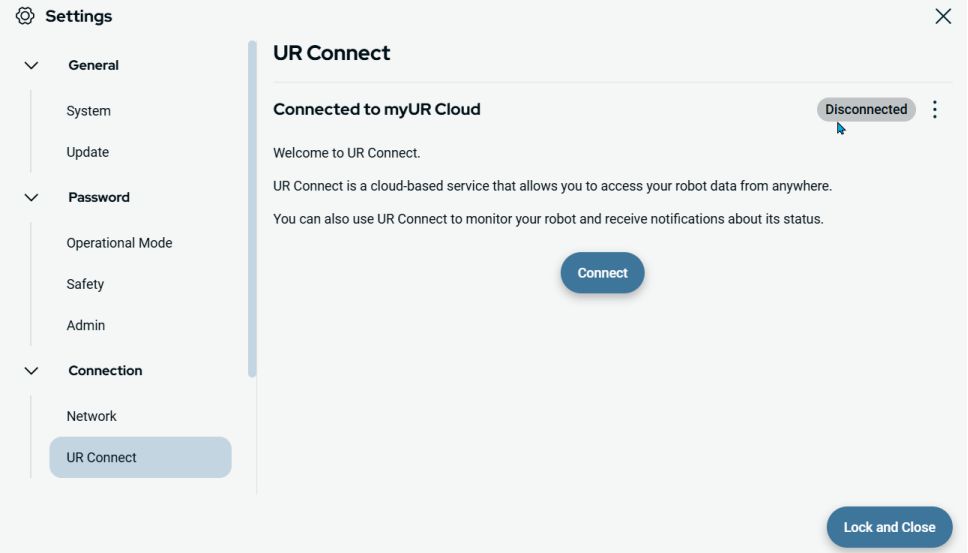
6.3.2. UR Connect

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.

6.4. Security

Description

In the security settings, you can access the following:

- Secure Shell (SSH) Access
- Permissions
- Services

6.4.1. 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. 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.

6.4.2. 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

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.

6.4.3. 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. 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

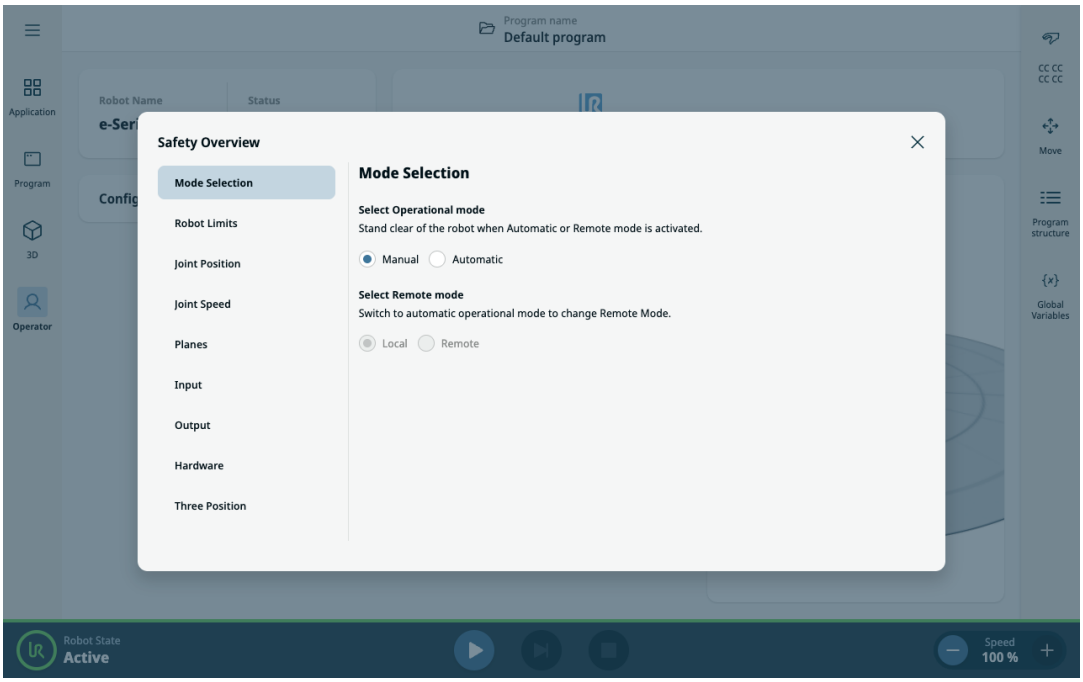
To switch between modes, in the Right Header, select the profile icon to display the Mode Selection.

- 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".



7.1. Manual Operational Mode

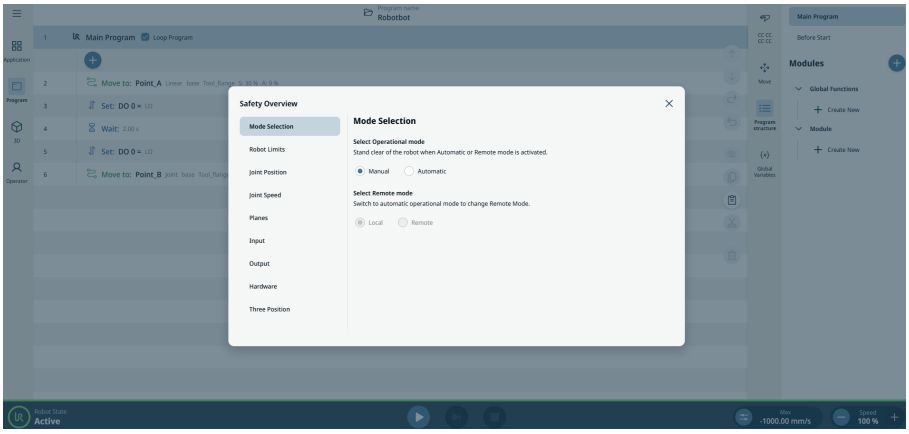
Description

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

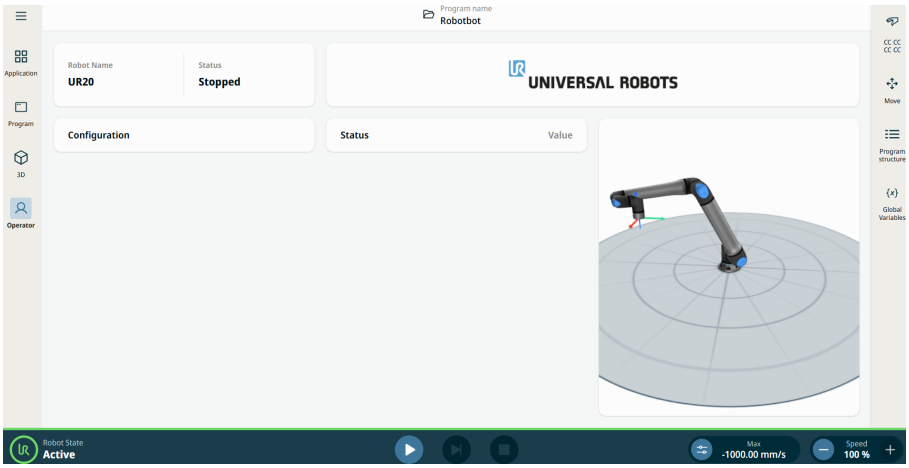
- A 3-position enabling (3PE) device must be held in its center position.
- Modify and save programs.
- Reduced speed control to 250 mm/s.
- Manual high speed is enabled by adjusting the **Manual High Speed** slider in the footer. A confirming action is required before higher speeds can be achieved.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



7.2. Automatic Operational Mode

Description

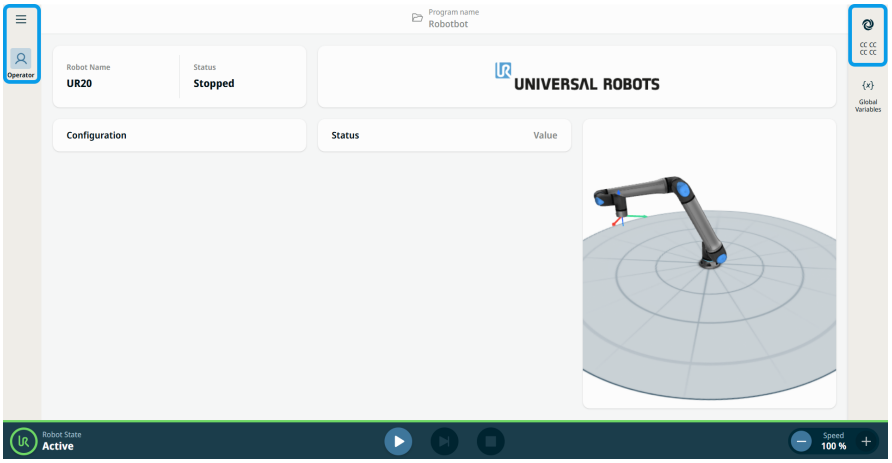
In **Automatic Mode**, the robot can only execute a program of predefined tasks. You cannot modify or save programs and installations. This has two properties:

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

To use Automatic mode

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

8. Remote Control

Description

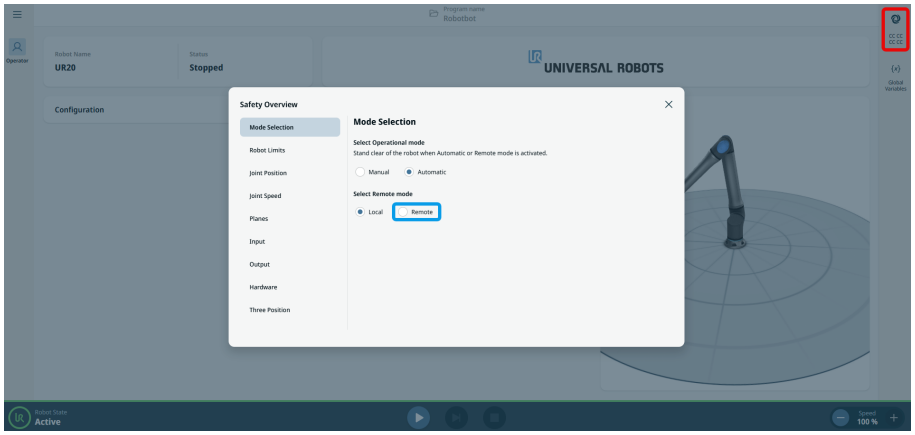
Remote Control allows you to control the robots via external sources, such as controller sockets, IOs, and the dashboard server. The other control is via the teach pendant (TP), called local control.
Remote control can be used to send simple commands to PolyScope X such as starting or loading programs, 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 no 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.

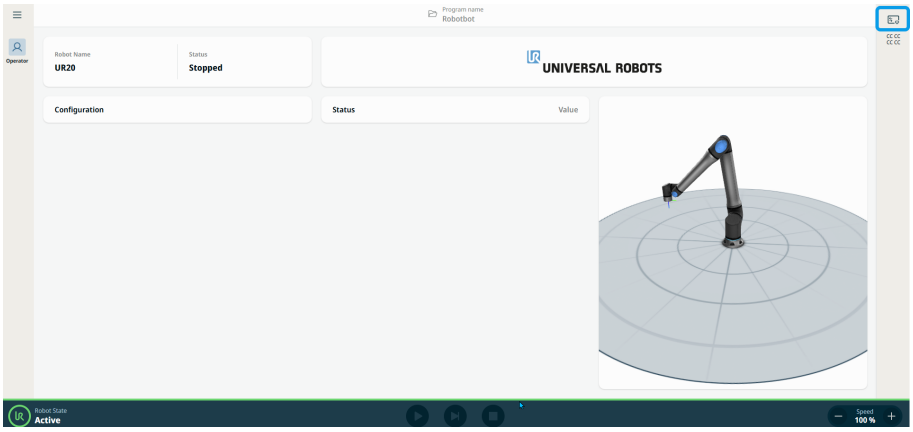


3. In the **Select Remote mode**, tick the Remote button.

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. 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.



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.

Settings Requirement

Control of the robot via network or digital input is, by default, restricted.

- Enable and select the Remote Control feature removes this restriction.
- Enable Remote Control by switching to the Local Control profile (PolyScope X control) of the robot, allowing all control of running programs and executing scripts to be performed remotely.
- Enable the Remote Control feature in Settings to access Remote mode and Local mode in the profile.

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

9. System Manager

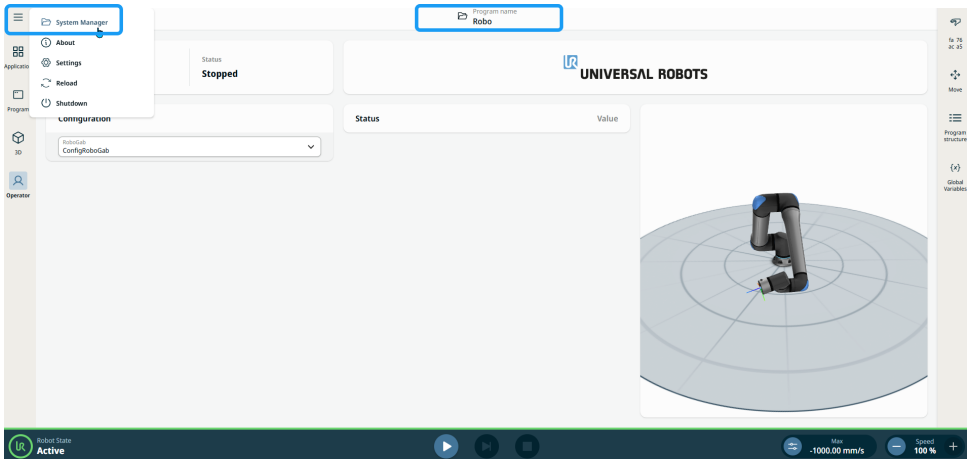
Description

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

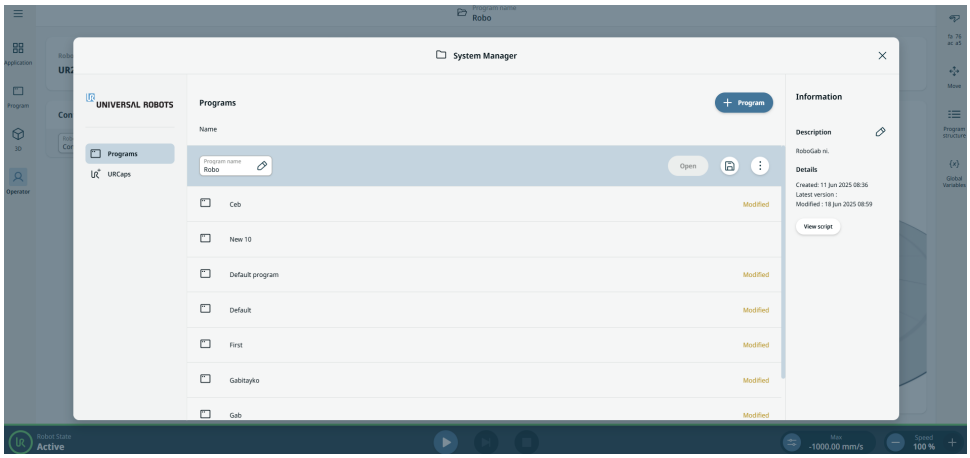
- See all created programs and installed URCaps
- Create a new program
- Import programs
- View script of selected program
- Write a program description

To access System Manager

There are two ways to access the **System Manager**: via header and via hamburger icon.
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.

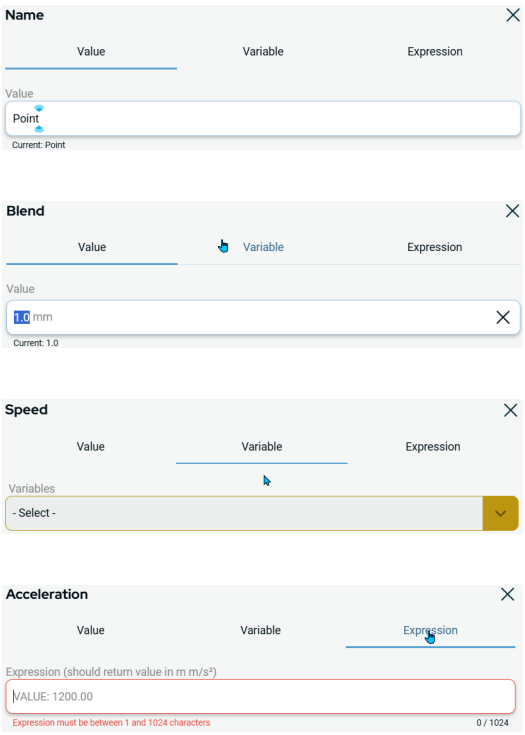


On the left column are found the **Program** section and **URCaps** section.
On the center column are found the **Program Manager**, which lists the created program, and the **+ Program** button/icon, which enables the creation of new program and importing a program.
On the right column are found the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

10. 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

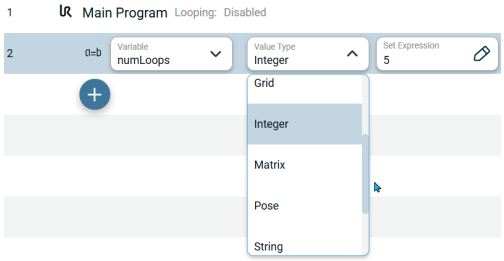
The **Value** tab is used when you know the exact fixed value of a program. It restricts you to values appropriate for the given variable.
The **Value Type** drop-down tab in the **Assignment** command contains the type of variable you are creating and has one of the following values:

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 to	p[0,0,0,0,0,0]
String	Text	"Hello World"
Waypoint	A waypoint as 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.

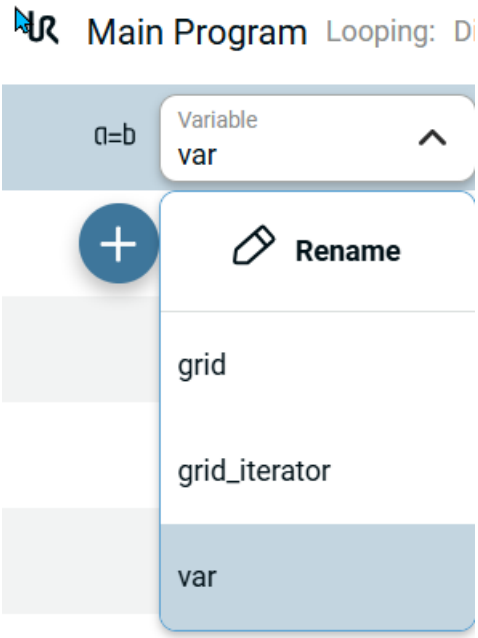
Variable tab description

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 or a Move To node, 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.

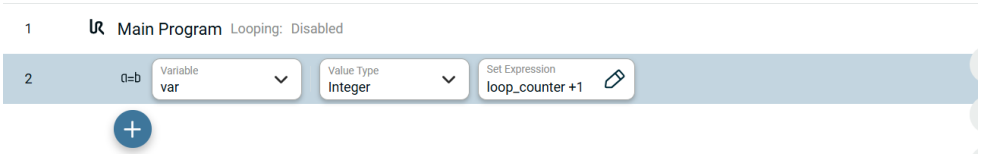
Expression tab description

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 left toolbar, 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 **Set Expression** field.
6. Tap the pop-up **Expression** tab and set the expression to **loop_counter +1**.
7. Tap **Confirm**.



11. OptiMove

Description

The **OptiMove** motion value is an advanced feature of PolyScope X **Move to** command. See Move To command.

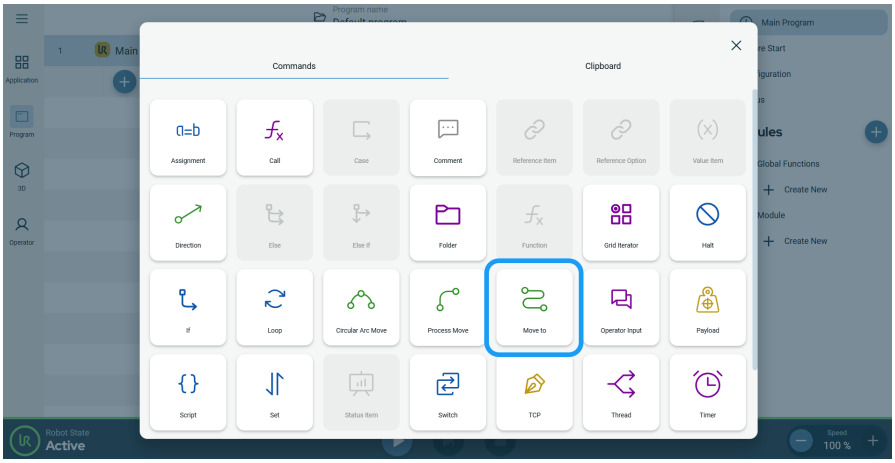
OptiMove allows you to have the following benefits:

- 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.

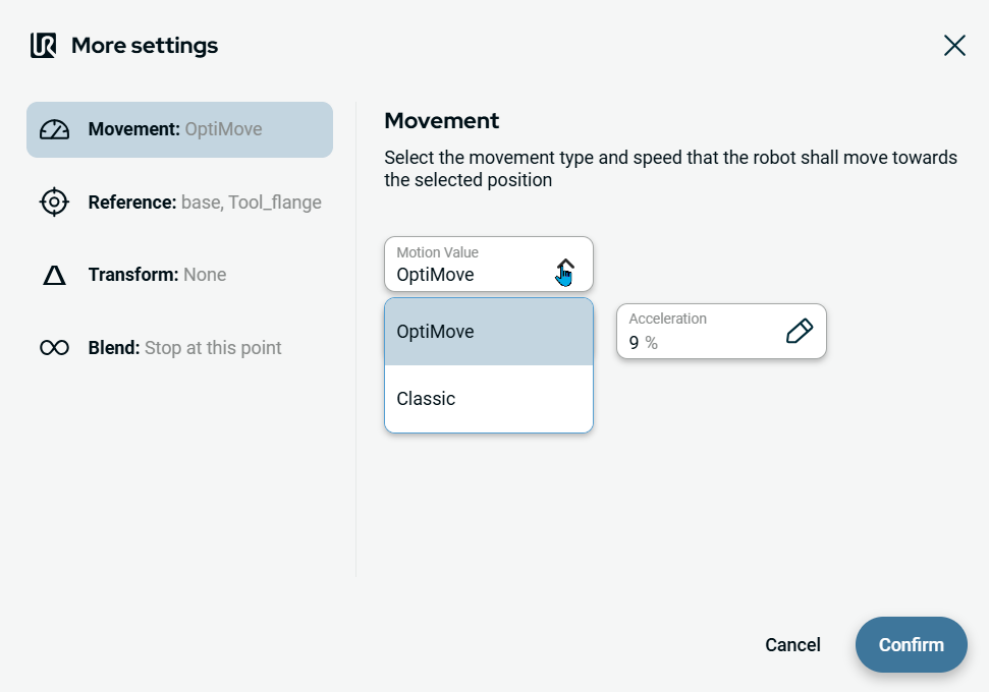
To access OptiMove

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



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

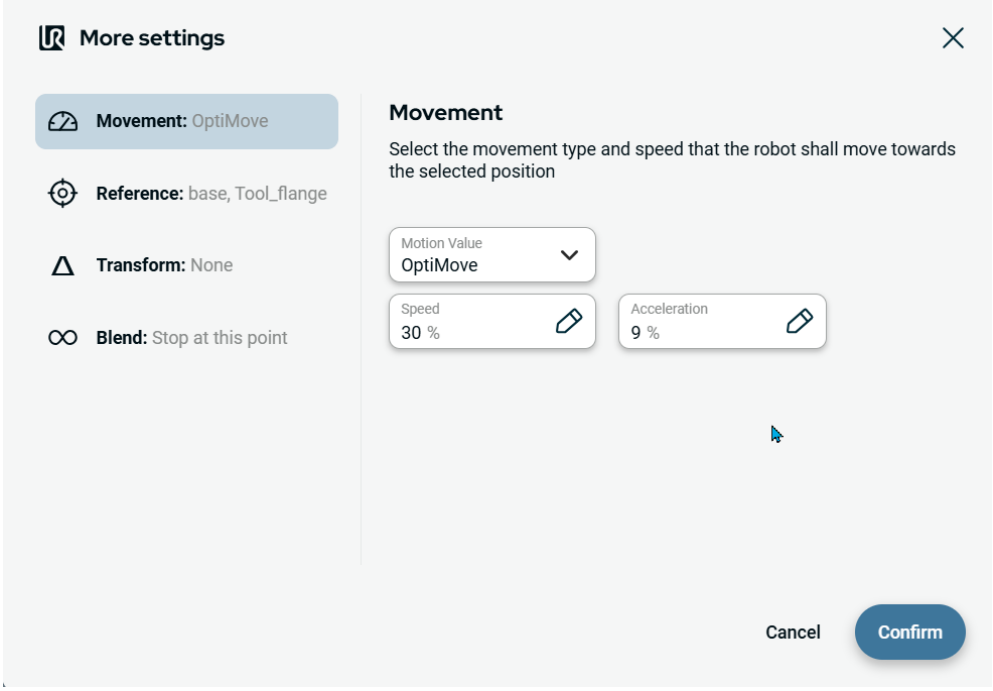
3. Tap the **More** button and the **More settings** screen appears.



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

   On the right side, two tabs and a field appear:

   - Motion Value field
   - 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.

12. 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.

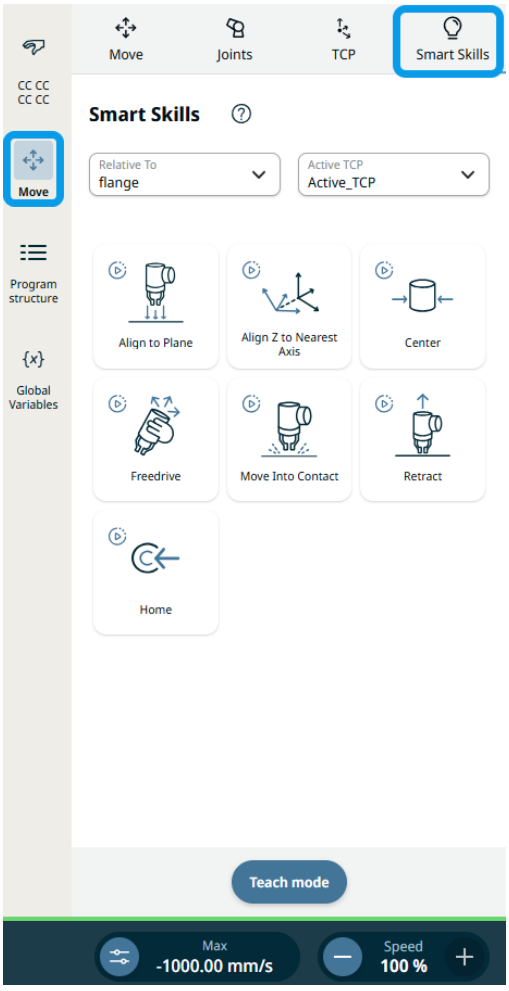
To access Smart Skills

There are two ways to access Smart Skills:

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

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.

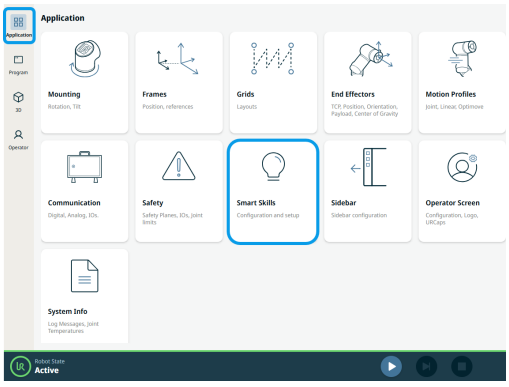


3. In the **Relative To** field, you can choose from the four features or frames or default coordinate systems:
   - **World**. This frame is the same as the base when the robot is mounted in a fixed location. The world frame is always fixed even when the robot is mounted in a moveable location, and 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.

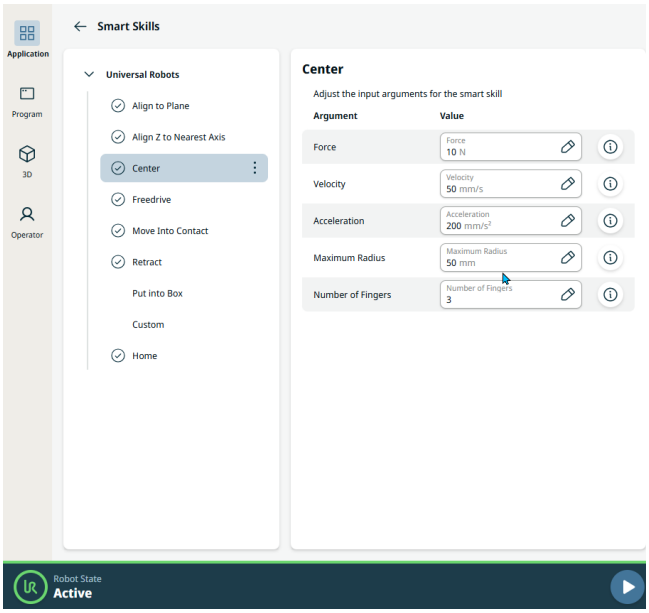
     See Frames to know more when to use the pertinent default coordinate system.
4. In the **Robot** field, you choose either **Active_TCP** or **Tool_flange**. If Active_TCP is selected, the name of the current active TCP is displayed.

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.
2. Choose the Smart Skills application menu.



You are directed to the Smart Skills application screen.



These are the seven Smart Skills to choose:

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

12.1. Align to Plane

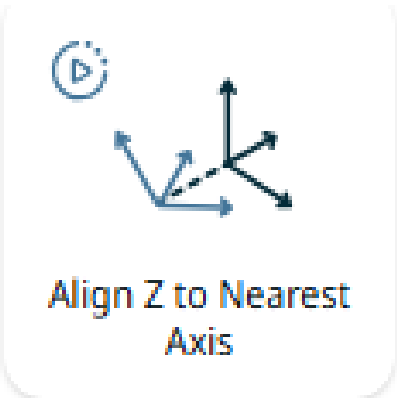
Description

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.

12.2. 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

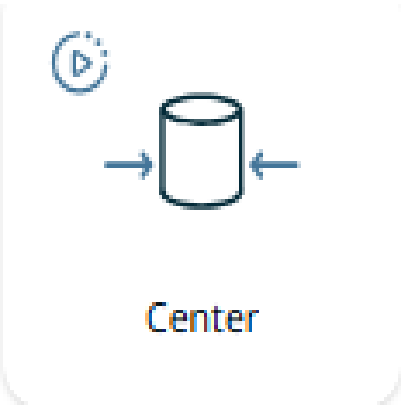
If you have a gripper and want to pick up an item, use this smart skill in order to have the robot actually move around in a predefined pattern in order to figure out where is the item positioned and what is the center of that. In this way, you can then center the robot.

12.3. 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



12.4. 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.

12.5. Home

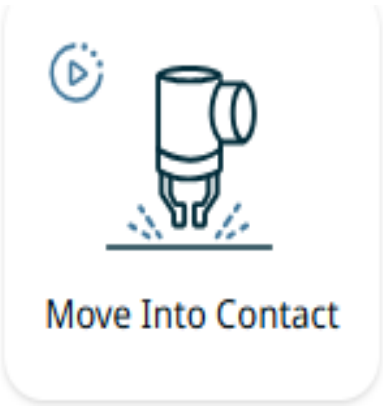
Description

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.

12.6. Move Into Contact

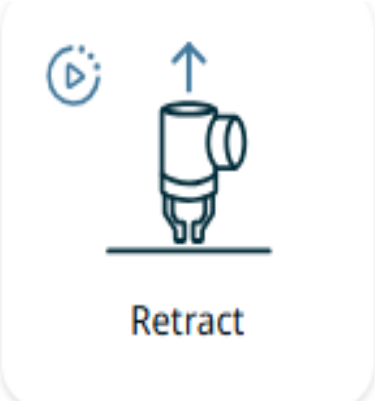
Description

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.



12.7. Retract

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



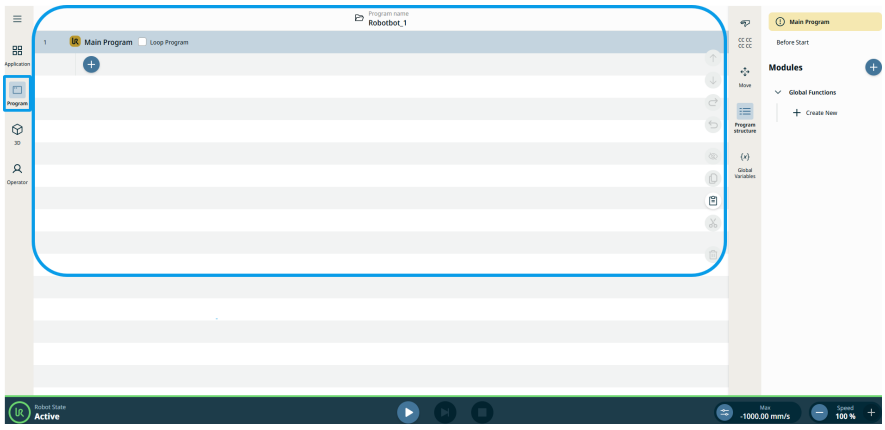
13. Program

Description

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

To access Main Program screen

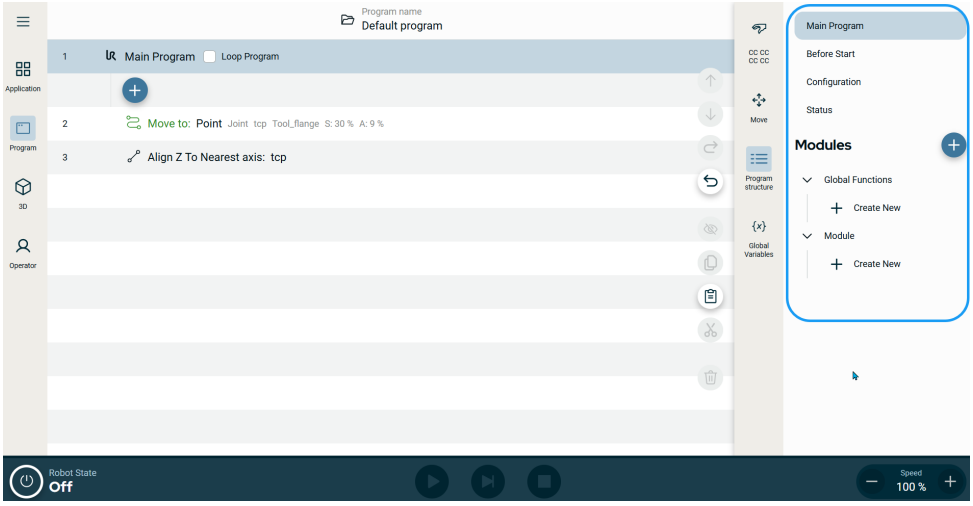
1. Tap the Program icon  on the main navigation.
2. You can see the **Main Program** setting in the main screen.



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

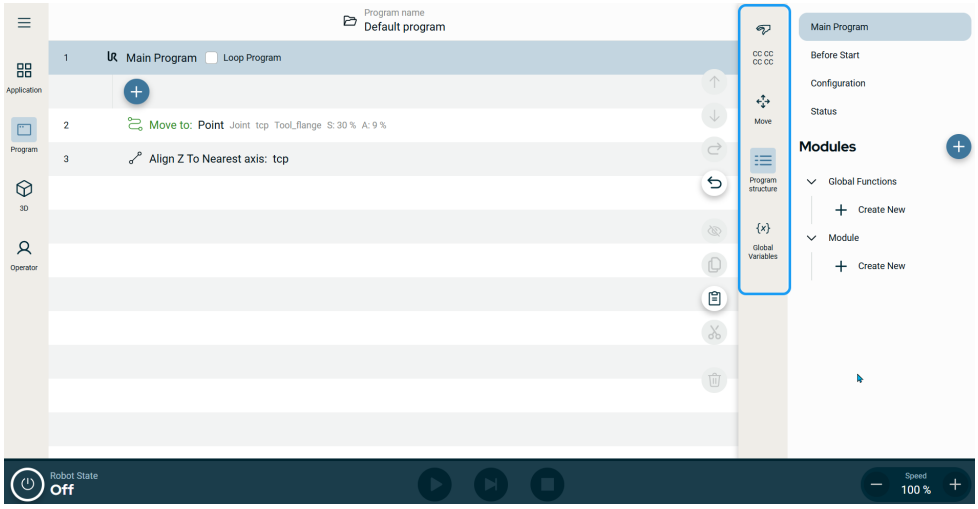
You can see the control tabs of the Main Program at the right side of the screen.



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.

13.1. Create Program

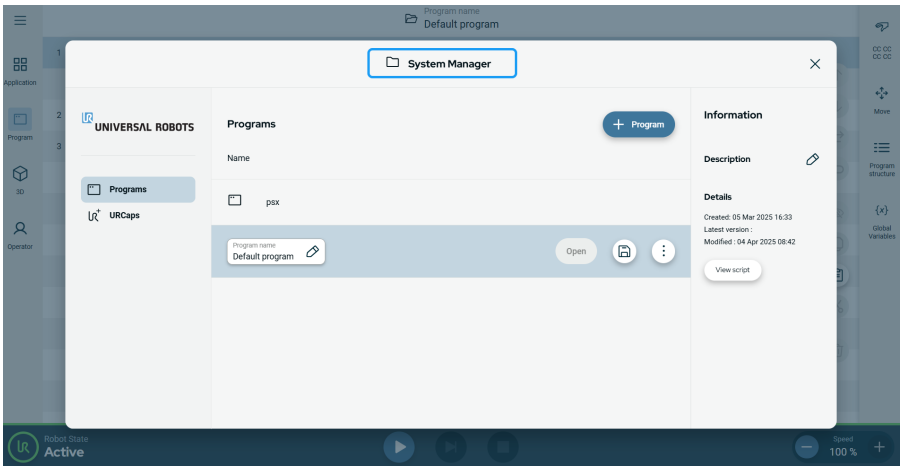
Description

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

To open Create program menu

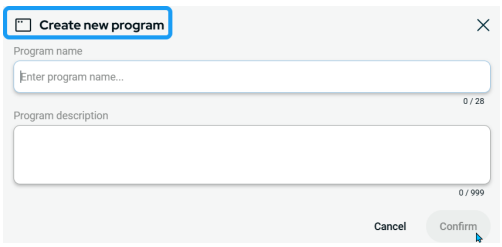
1. Tap the **Operator** icon  on the left toolbar.
2. Tap the **Program name** tab found topmost in the main screen.

   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**.

13.2. Modify Program

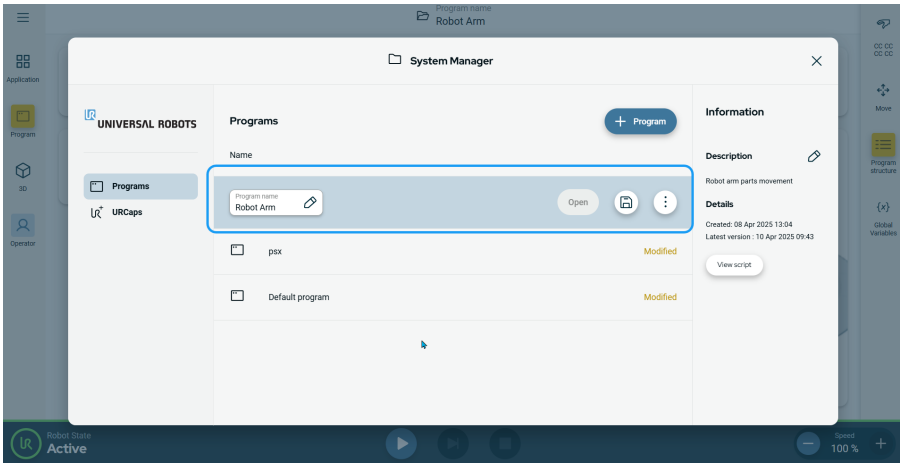
Description

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.



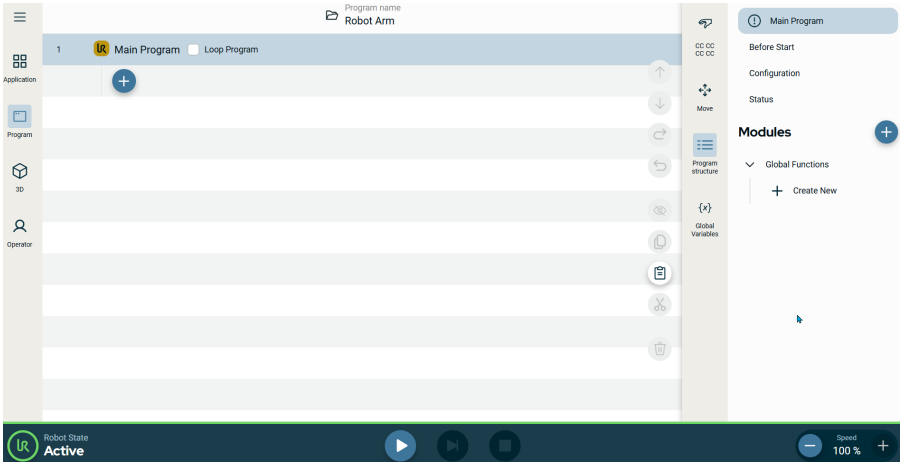
   The tab contains four buttons:

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

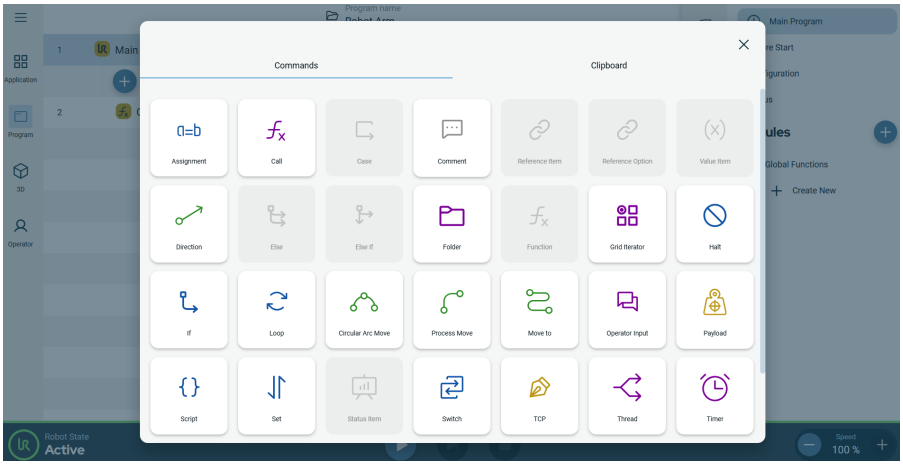
   You can see the details of the date created and the latest version of the program on the right side of the screen. A button to view the script is also provided.
3. Tap the **Program name** button to edit the name and **Confirm**.

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**

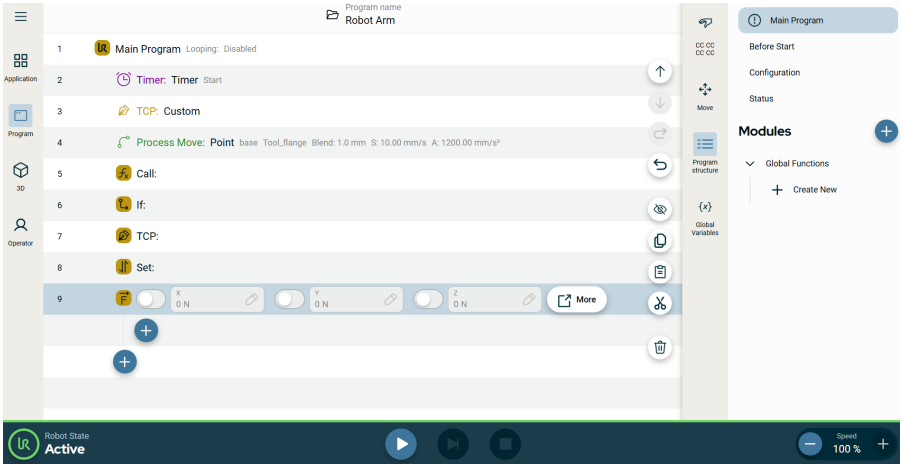
   Further to the right of the sidebar, you can access:
   - **Main Program**
   - **Before Start**
   - **Modules tree**



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**

13.3. Command Nodes

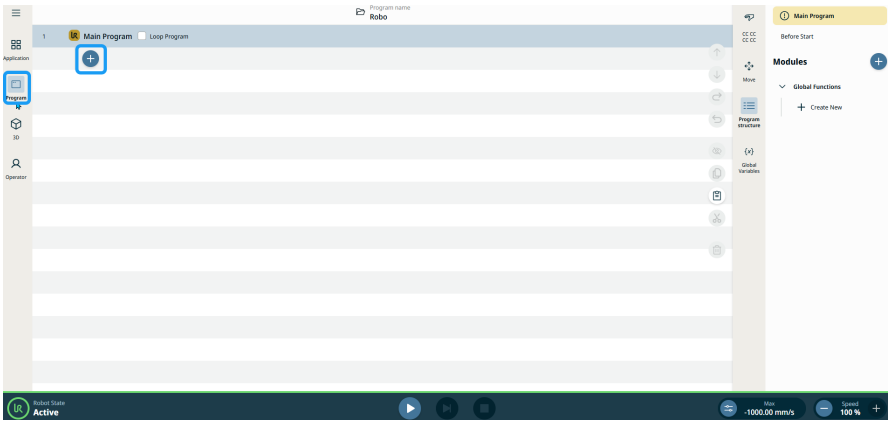
Description

The PolyScope X command nodes contain the commands that robot users apply to create a program in the simplest way.
The nodes are 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.

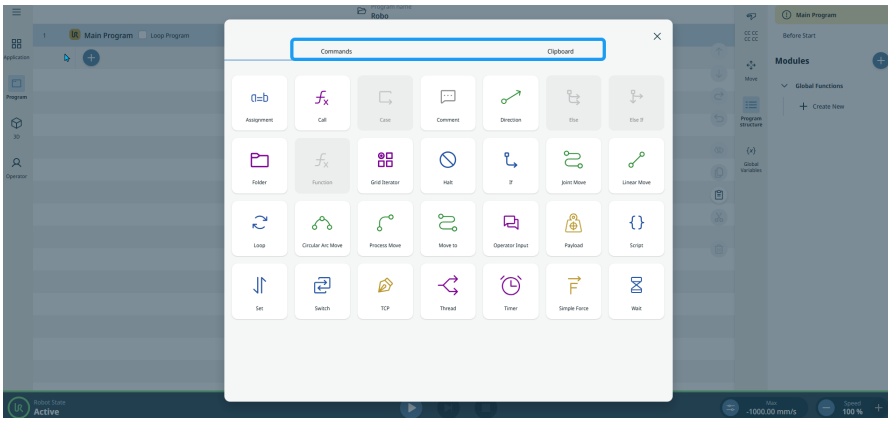
To access Command nodes

1. Tap **Program** on the left toolbar and then the **Add** icon in the main screen.

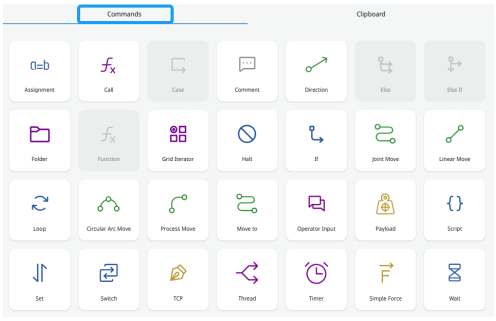


A new screen appears with two toolboxes:

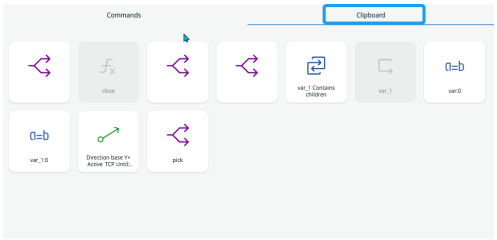
- **Commands**. Contains the 28 command nodes, which are categorized by colors.
- **Clipboard**. Contains all cut-and-copied commands used throughout in creating a program.



2. By default, the **Commands** toolbox is what appears when you open the program menu.



3. Tap the command node you want to access.
4. Tap the **Clipboard** toolbox, and you can see and review the history of cut-and-copied commands used.

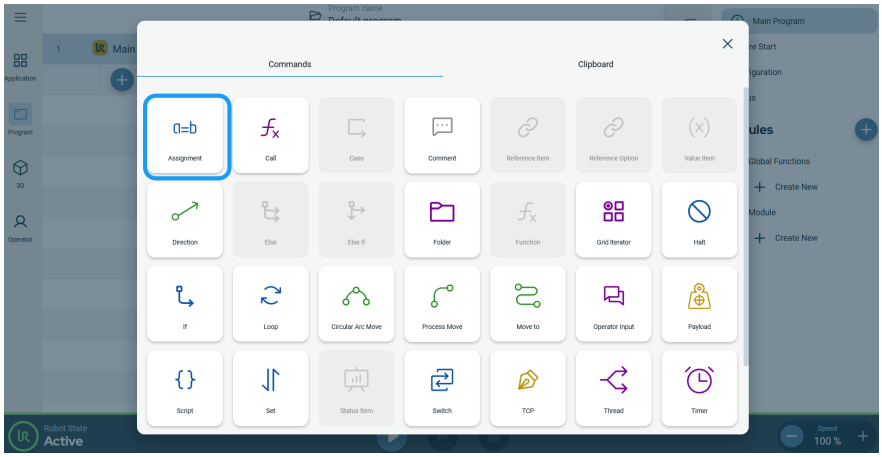


13.3.1. Assignment

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

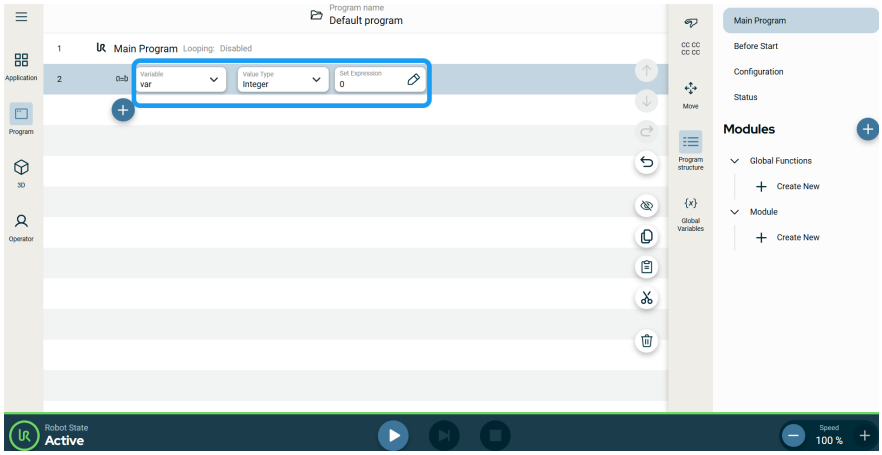
To access Assignment command

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
- Set 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.

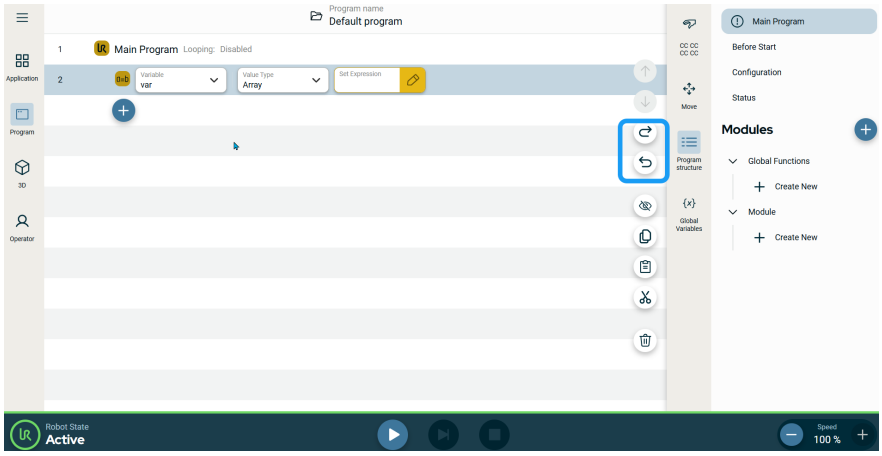
Type	Allowed values / examples	Examples
Array	A list of values	
Boolean	True / False	True
Float	A decimal point value	1.23
Grid	A grid defined in the application	
Integer	A whole number	5
Matrix		
Pose	A pose for the robot to move to	p[0,0,0,0,0,0]
String	Text	"Hello World"
Waypoint	A waypoint as defined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

5. Tap the **Set Expression** field and select your variables and expression.

   The Set Expression is the value to assign the variable.

6. Tap **Confirm**.

   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.



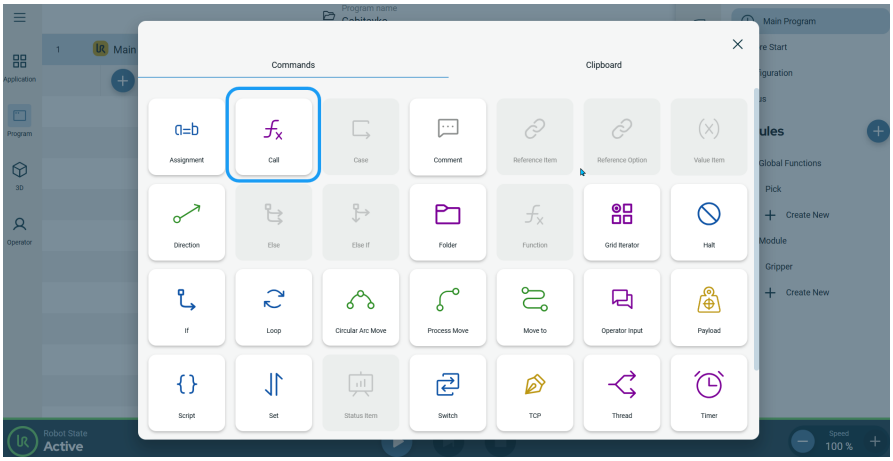
13.3.2. Call

Description

The **Call** command enables you to execute and run created global functions and modules, which are seen in the multitask screen.

To access Call command

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



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

3. Tap the **Module** field.

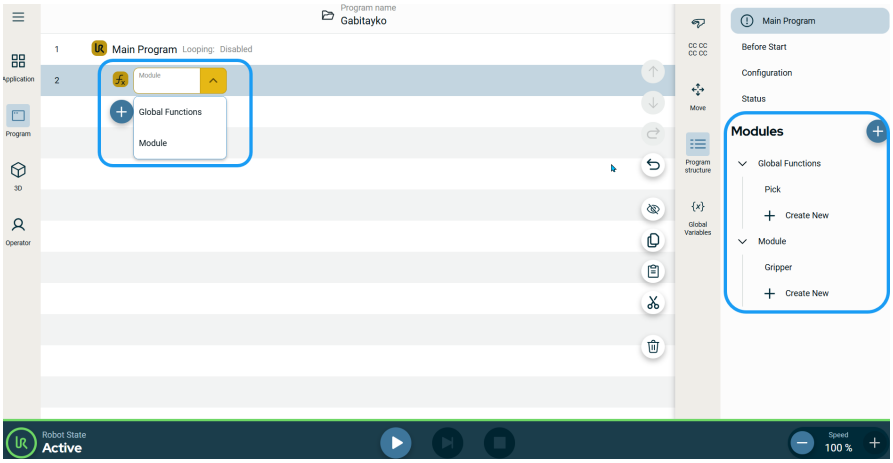
   Two options are available:

   - Global Functions

     Contains a list of functions for the selected module or the global functions, which appears in the multitask screen.

   - Module

     Contains a list of modules you have created, as well as the Global Functions option for those functions at global scope, which appears in the multitask screen.



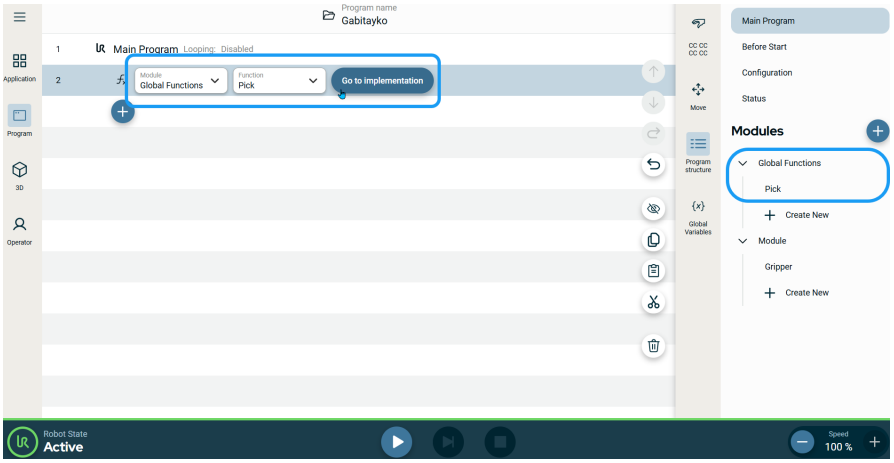
To use Global Functions

4. Tap **Global Functions**.

   A **Function** field is inserted in the right side.

5. Select the appropriate global function to call.

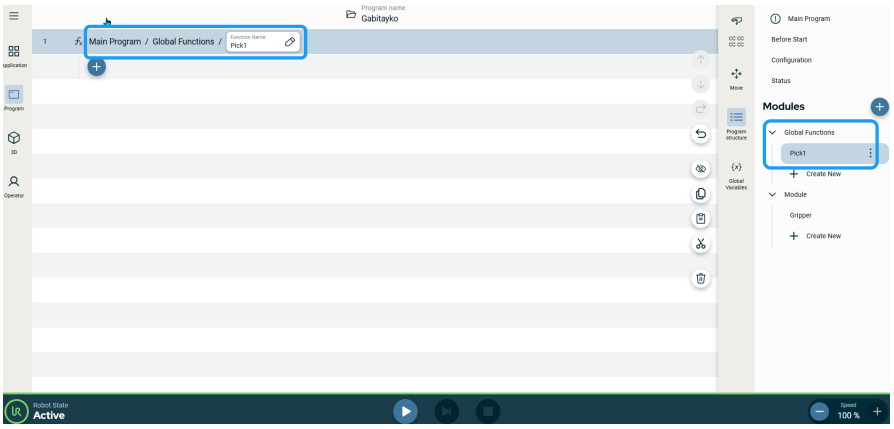
   A **Go to implementation** tab appears in the right side.



6. Tap the **Go to Implementation** tab.

   The Global Functions is then inserted into the Main Program.

7. If you want to rename the called function, tap the **Function Name** field and edit. The edited name also appears in the multitask screen.



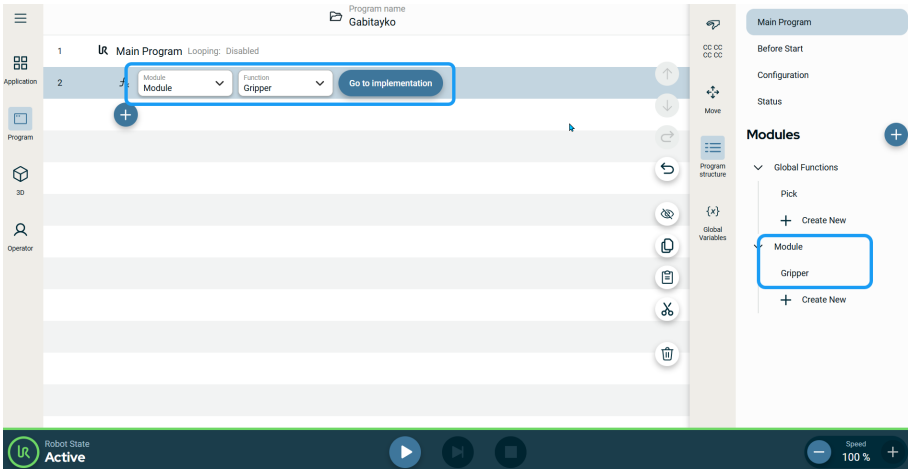
To use Module

8. Tap the **Module** field and select **Module**.

   A **Function** field is inserted in the right side.

9. Select the appropriate module to call.

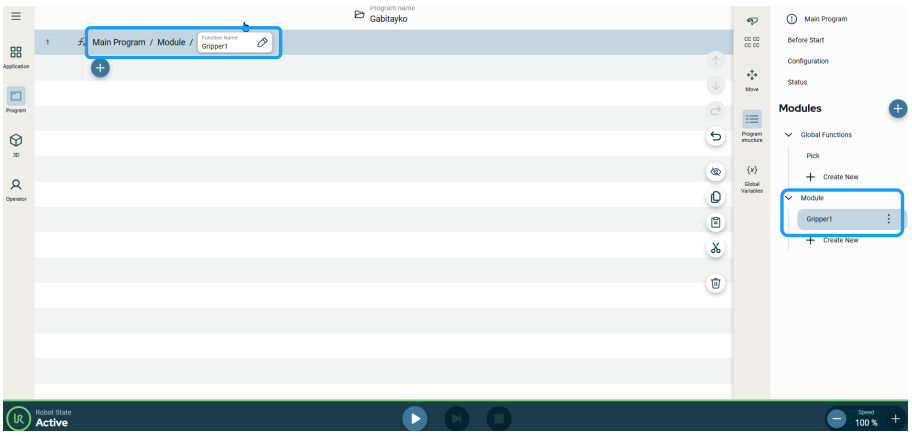
   A **Go to implementation** tab appears in the right side.



10. Tap the **Go to Implementation** tab.

    The Module is then inserted into the Main Program.

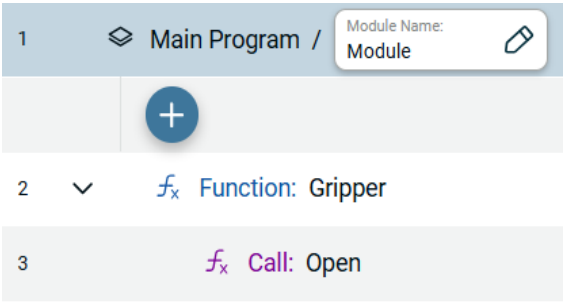
11. If you want to rename the called module/function, tap the **Function Name** field and edit. The edited name also appears in the multitask screen.



Example **Opening a Gripper**

1. In the multitask screen, tap **Create New** under the Module tree and name it **Gripper**.
2. Tap **Create New** under the Global Functions tree and name it **Open**.
3. Go to **To use Module** for iteration.
4. Tap the add icon under the Module program tree.
5. Tap the **Call** icon in the Commands toolbox.
6. Go to **To use Global Functions** for iteration.

   This is now the program tree.

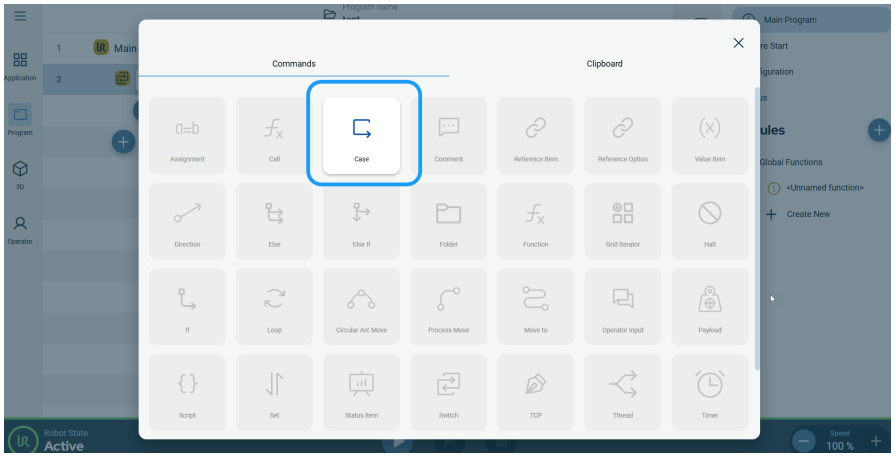


13.3.3. Case

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

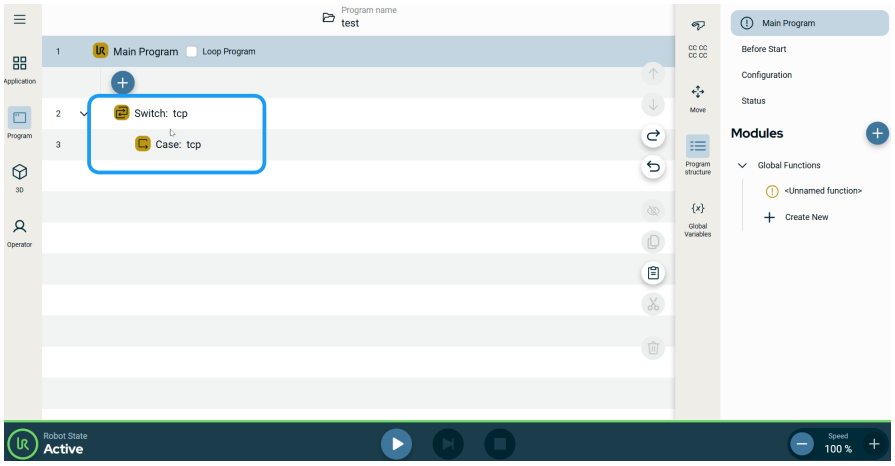
1. Go to the main program screen. See Command Nodes.
2. Access the Switch command.
3. After the input in **Set switch expression**, tap the first add icon from the child node of the switch command.
4. Tap **Case** icon in the Commands toolbox.



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.



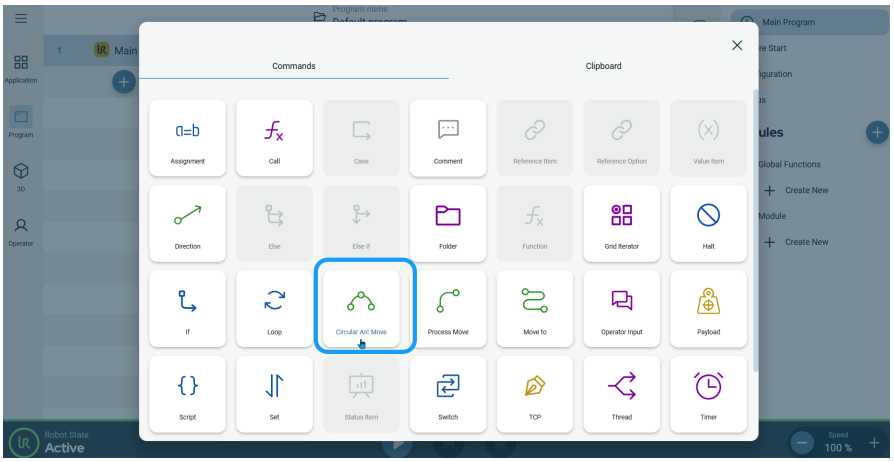
13.3.4. Circular Arc Move

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.

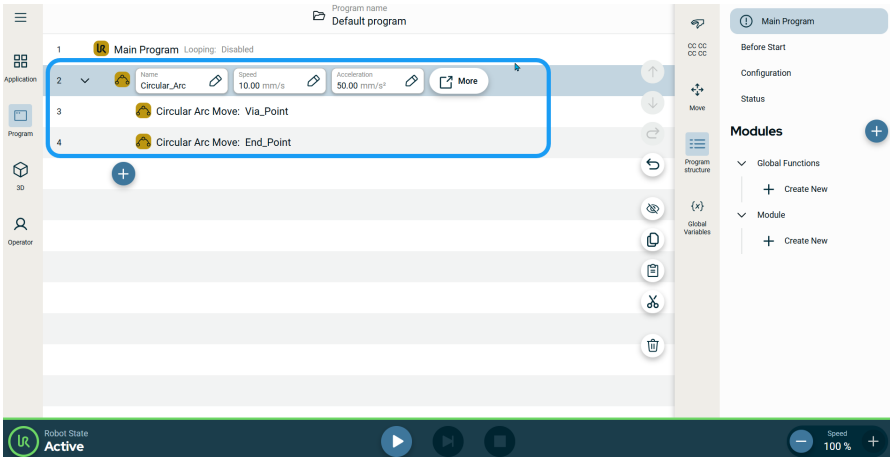
To access Circular Arc Move command

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.

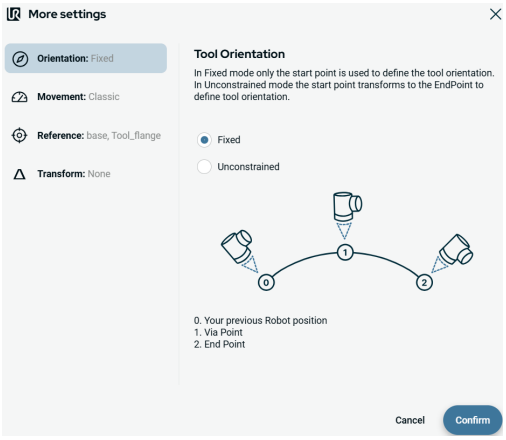
3. Tap the **Name** field, and three tabs appear:
   - Value
   - Variable
   - Expression
4. Enter the data in each tab and tap **Confirm**. See Value-Variable-Expression Tabs.
5. Tap the **Speed** field, and three tabs appear:
   - Value
   - Variable
   - Expression
6. Repeat step 4.
7. Tap the **Acceleration**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

To access More settings

1. Tap the **More** button, and the **More settings** screen appears.

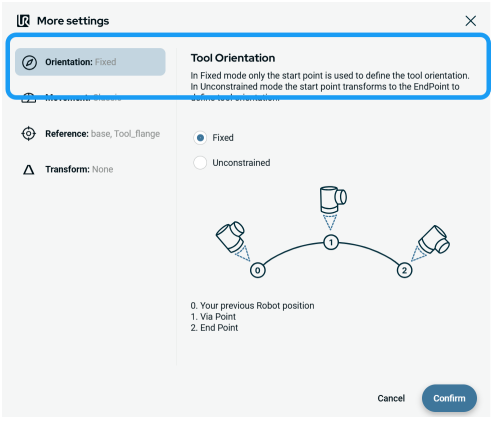
   On the left side, you see four advanced options:

   - Orientation
   - Movement
   - Reference
   - Transform



To use Orientation

1. Tap **Orientation**.



On the right side, two buttons are provided:

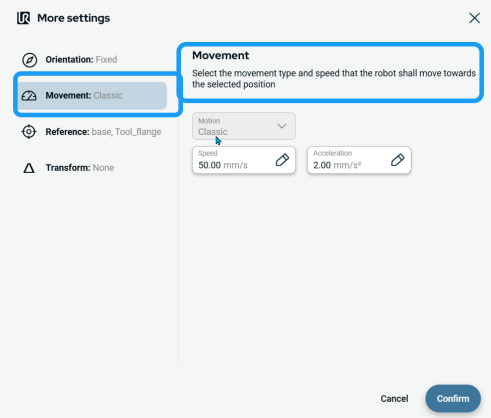
- Fixed
- Unconstrained

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

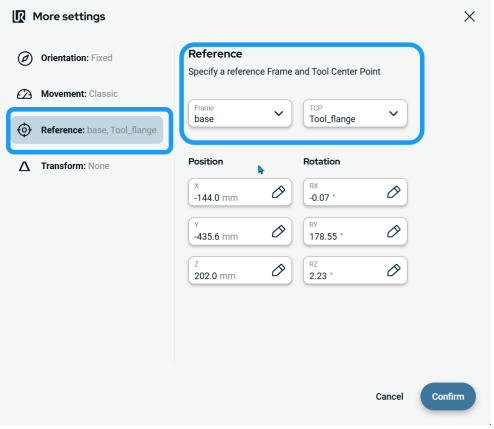
1. Tap **Movement**.



   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

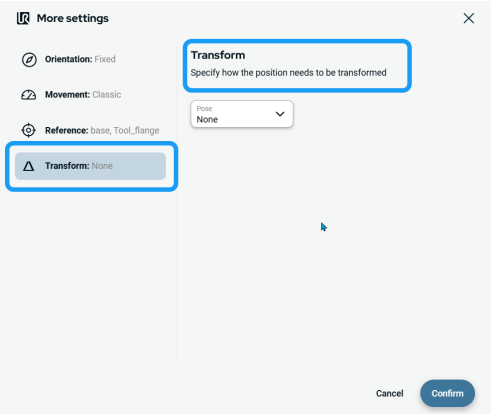
1. Tap **Reference**, and two editable **Frame** and **TCP** fields appear.



   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

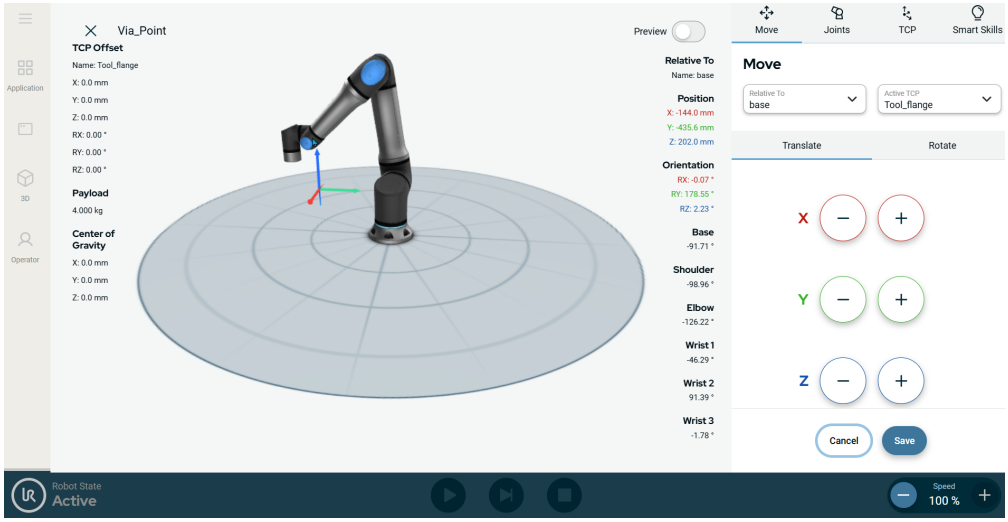
1. Tap **Transform**.
2. Tap the **Pose** field to specify how the position needs to be transformed and **Confirm**.



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**.

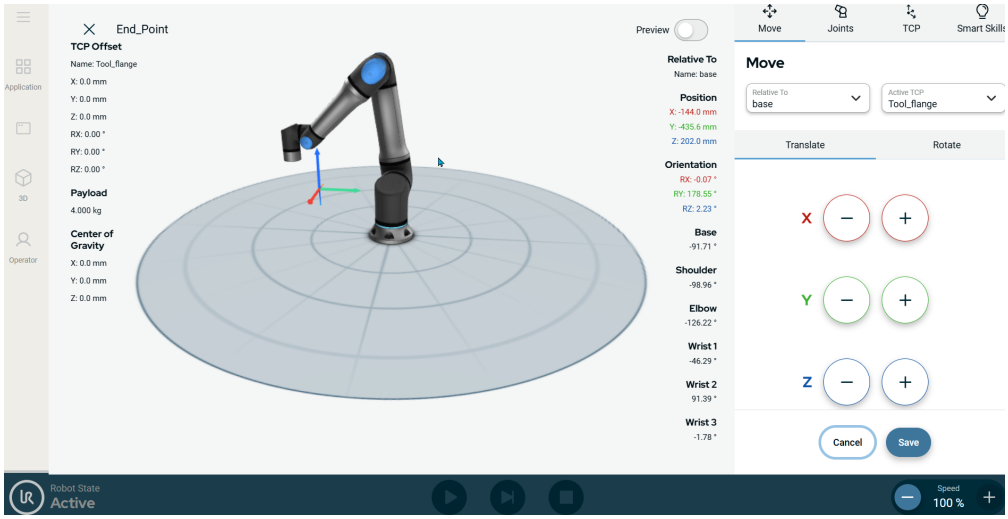
   The 3D viewer screen of the robot appears. (See Move To for details of use.)



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 Move To for details of use.)



13.3.5. Comment

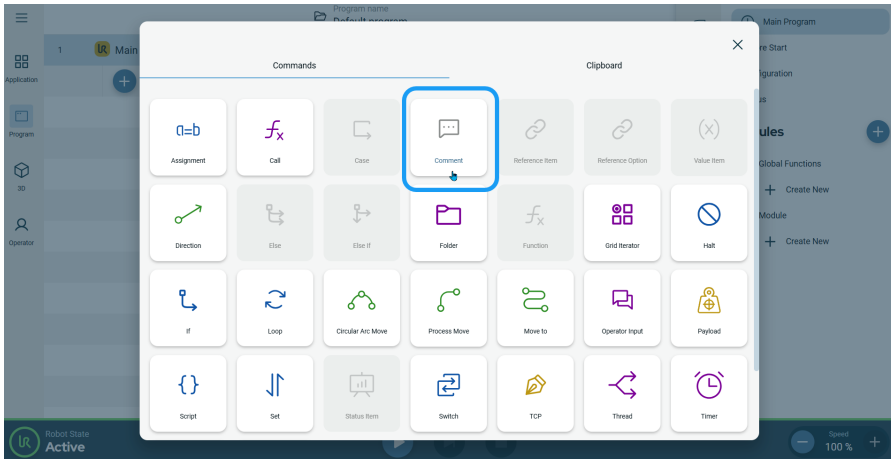
Description

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**.

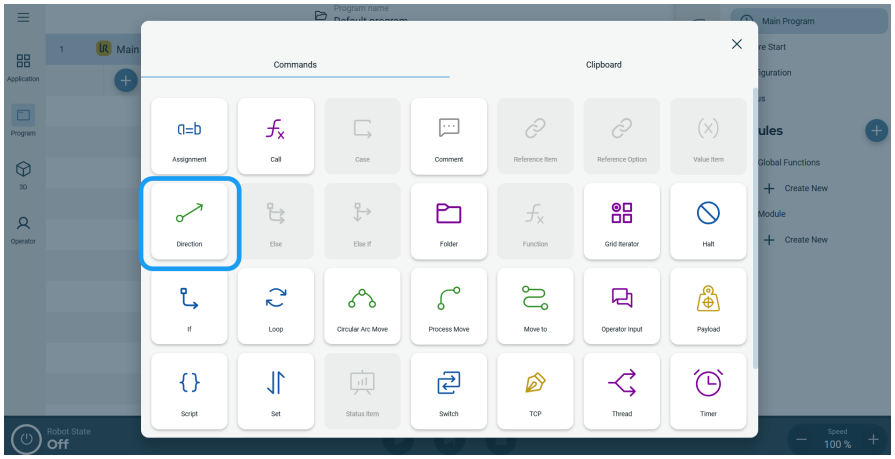
13.3.6. Direction

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.

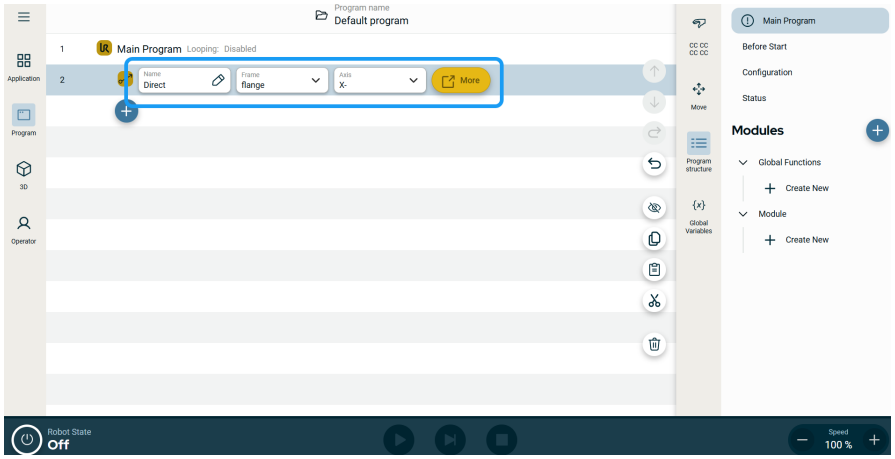
To access Direction command

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



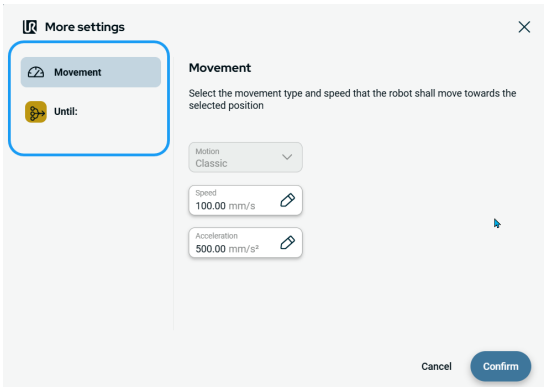
3. Tap the **Name** field.
4. Enter the Value name in the tab and tap **Confirm**.

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.

To use the More options

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

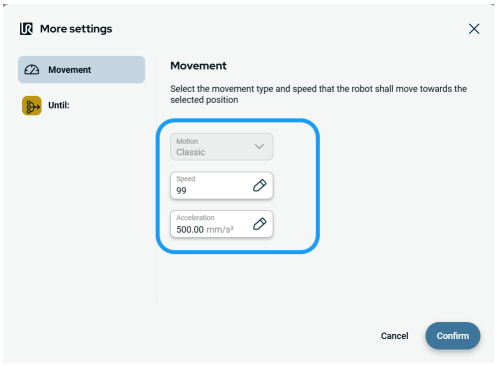
- Movement
- Until



To use Movement setting

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

   Note: The Motion field is inactive.



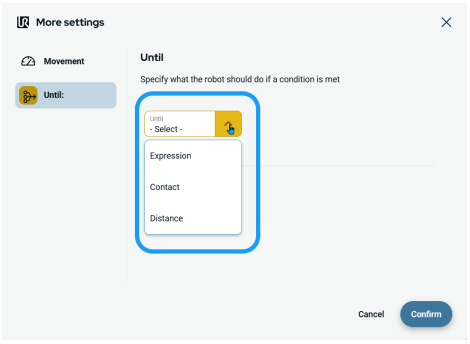
2. Tap the **Speed** field, and three tabs are available:
   - Value
   - Variable
   - Expression

   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.

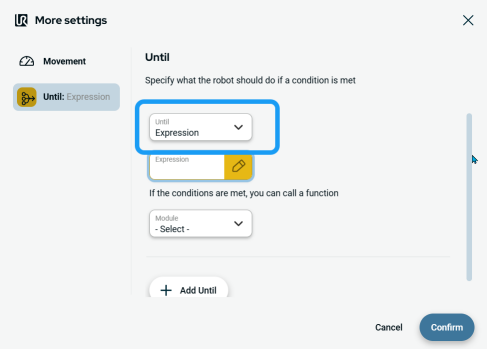
To use Until setting

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**.

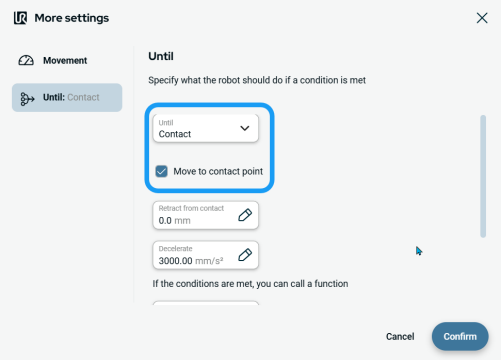
   An additional **Function** field appears to the right.
5. Tap **Add Until** button to add further conditions, and **Confirm**.

   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.

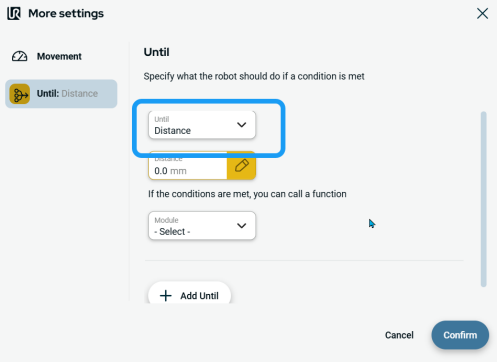
   Note: When you uncheck the **Move to contact point** box, the **Retract from contact** field becomes inactive.
5. Tap the **Decelerate** field, and three tabs are available:
   - Value
   - Variable
   - Expression

   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**.

13.3.7. Else

Description

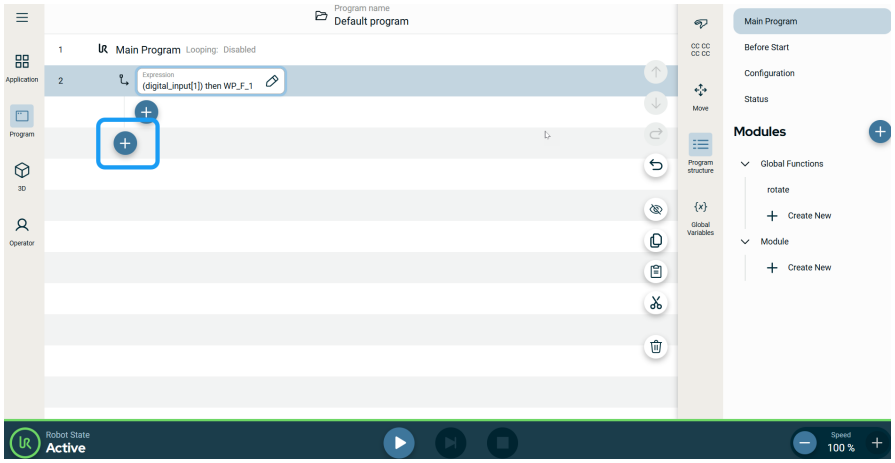
The **Else** command, in addition to Else If, is used together with the If command. It enables you to make statements to change the robot's behavior based on sensor inputs or variable values.
It is only insertable directly after an If or Else If node.
The children of this node are only executed if the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

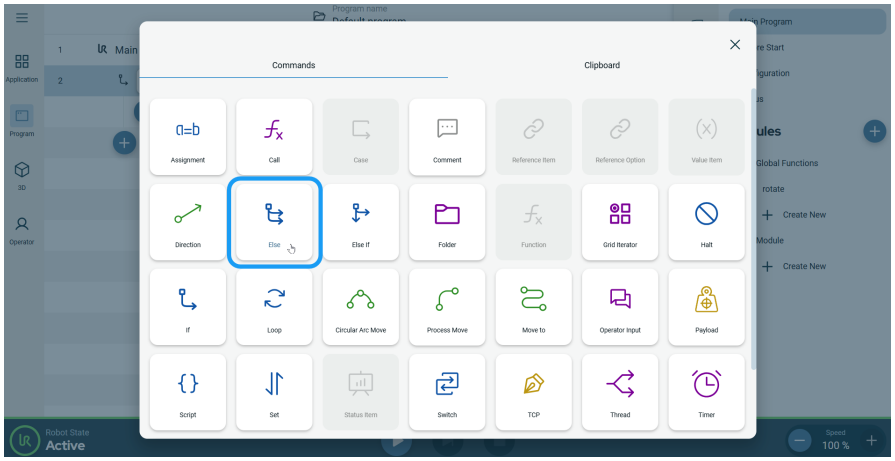
   Follow the process when accessing the If program. See If for more.

   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**.



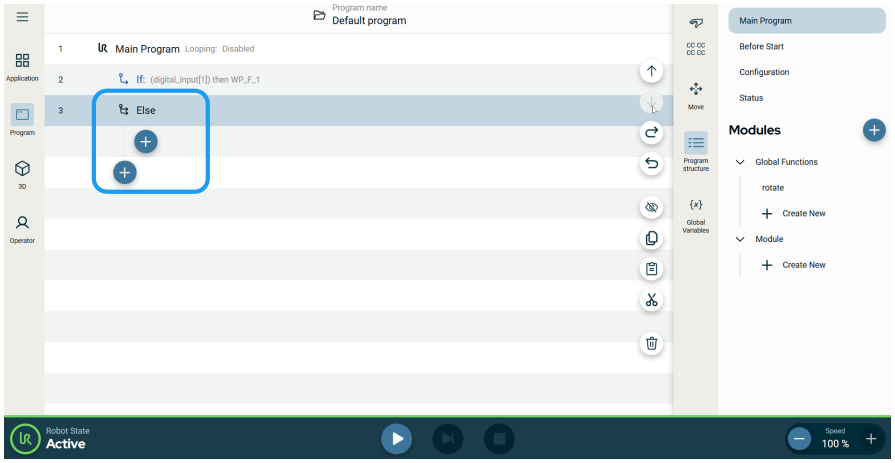
The Commands toolbox appears.

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.



13.3.8. Else If

Description

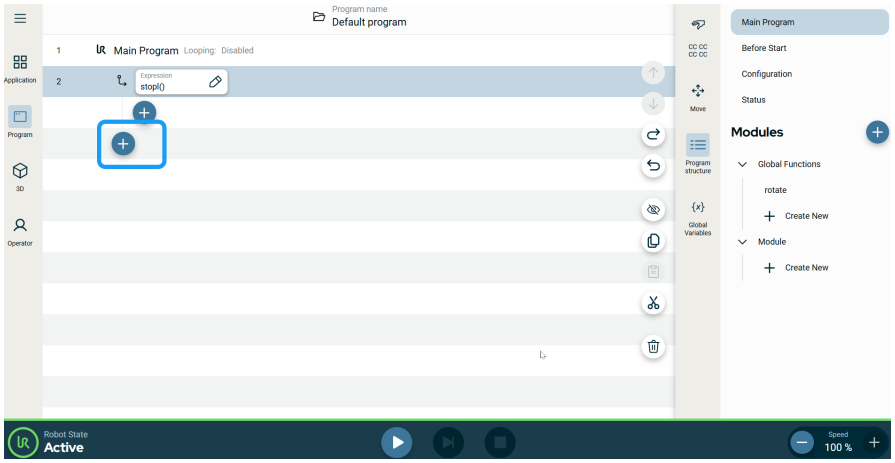
The **Else If** command, in addition to Else, is used together with the If command. It enables you to make statements to change the robot's behavior based on sensor inputs or variable values.
The Else If node is only insertable directly after an If or Else node.
The children of this node are only executed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

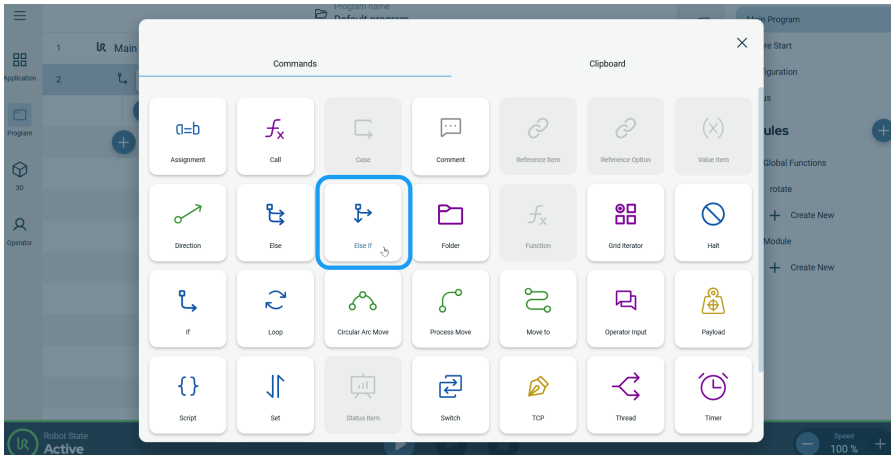
   Follow the process when accessing the If program. See If for more.

   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**.



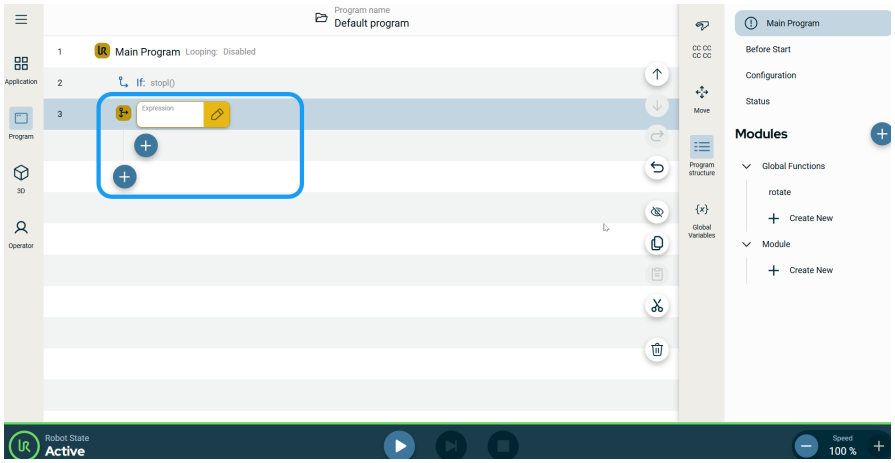
The Commands toolbox appear.

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.



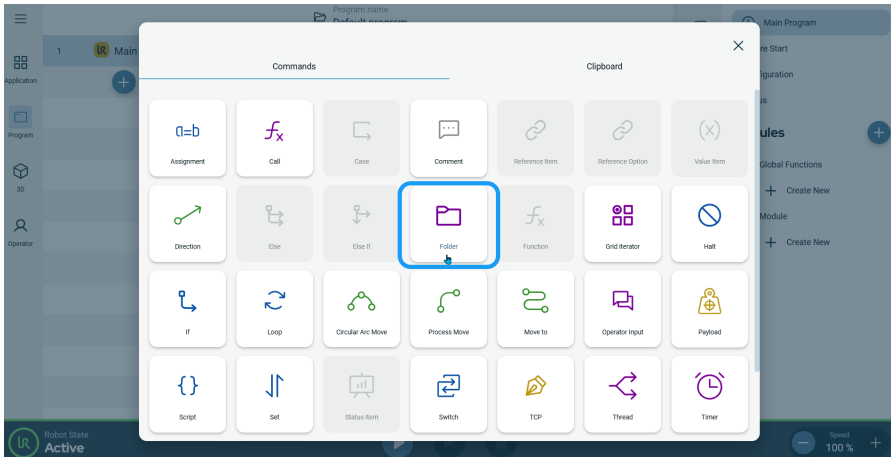
6. Tap the **Expression** field.
7. Create your expression .
8. Tap **Confirm**.
9. Choose and tap the appropriate **add icon** to access other commands for the robot movement.

13.3.9. 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.



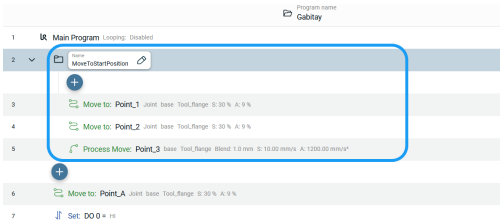
A Folder node, which is the **Name** field, is inserted into the program tree.

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.



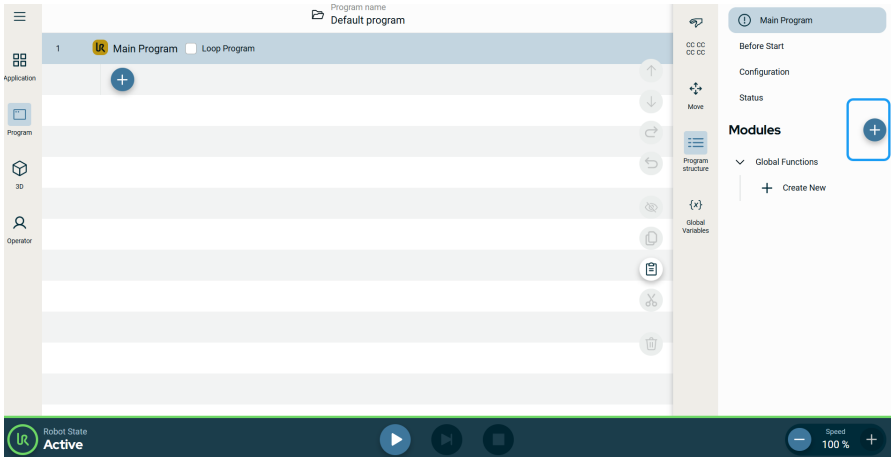
13.3.10. Function

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.

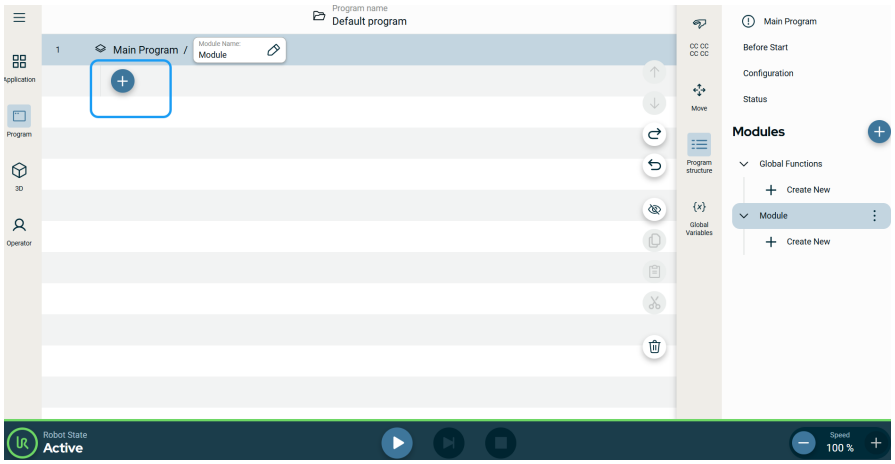
To access Function command

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

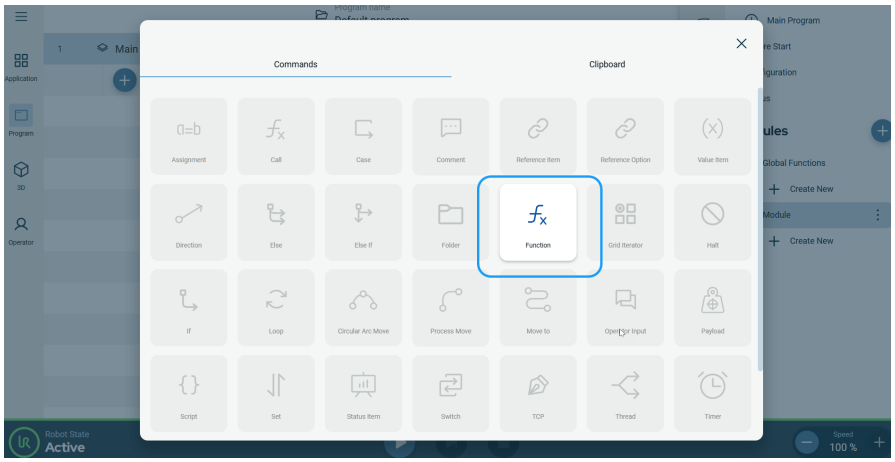


A new module is inserted into the module tree.

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**.

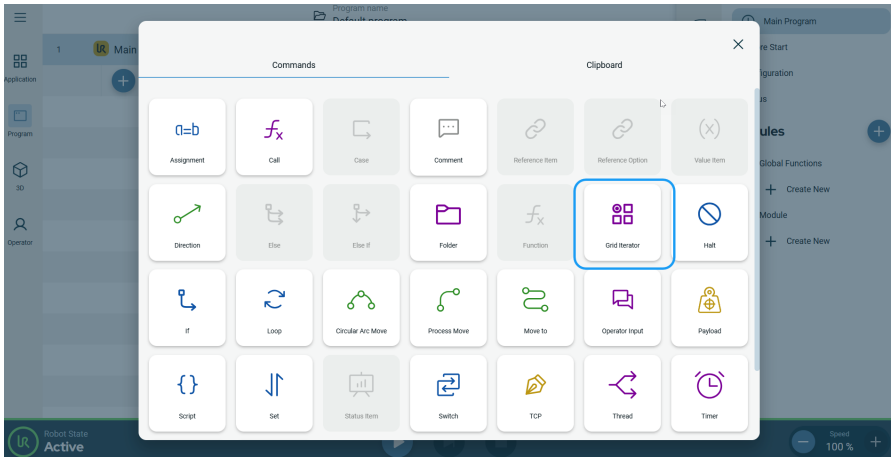
13.3.11. Grid Iterator

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.

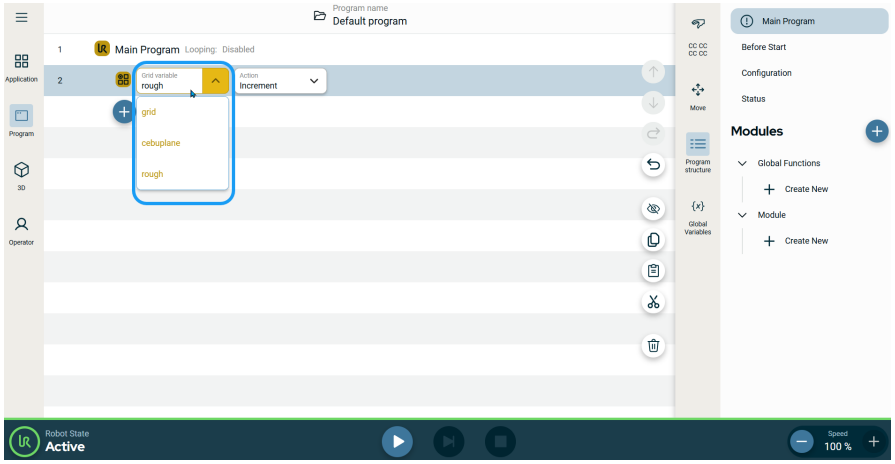
To access Grid Iterator command

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**.

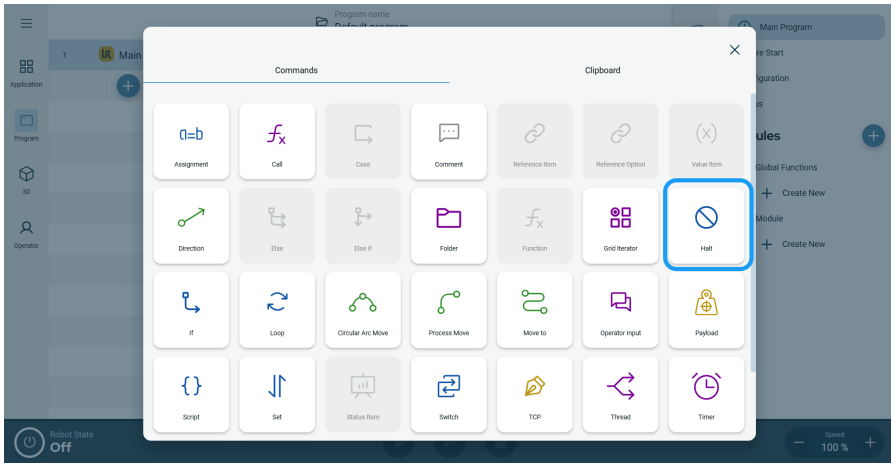
Each increment moves the variable to the next grid position. The pose variable, **grid_iterator**, can be used as the target in a Move node instead of a fixed waypoint, enabling sequential motion through the grid layout.

13.3.12. 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."

13.3.13. If

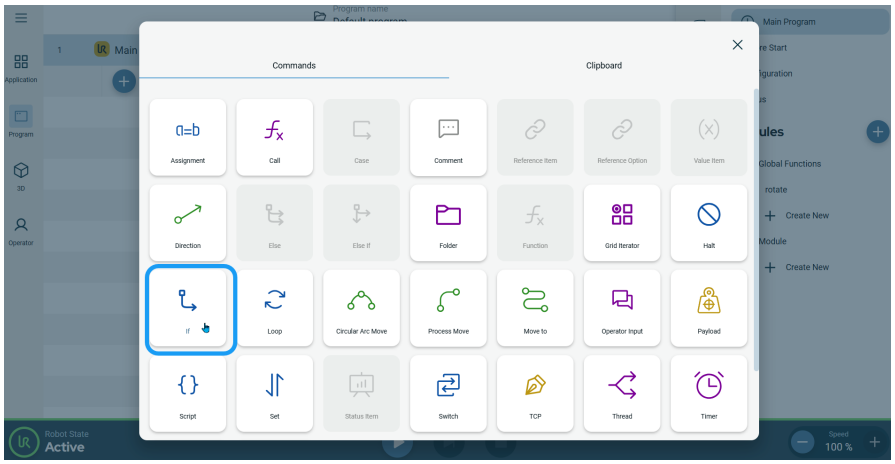
Description

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**.

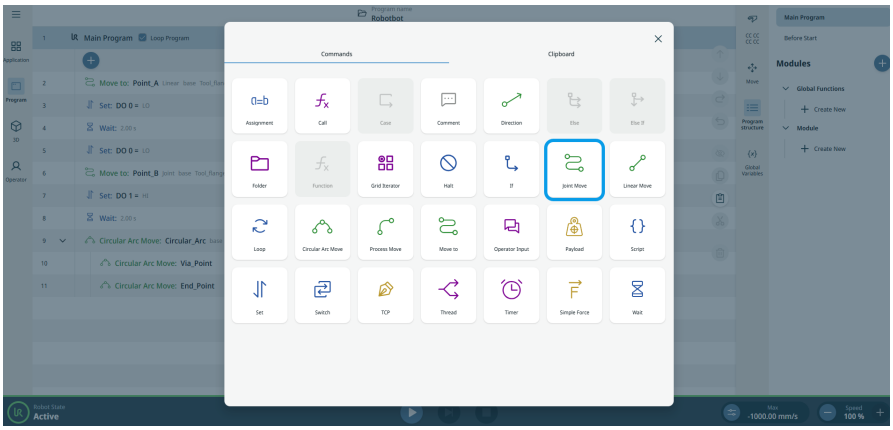
13.3.14. Joint Move

Description

The **Joint Move** works in a similar way to Move to command, but it 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.

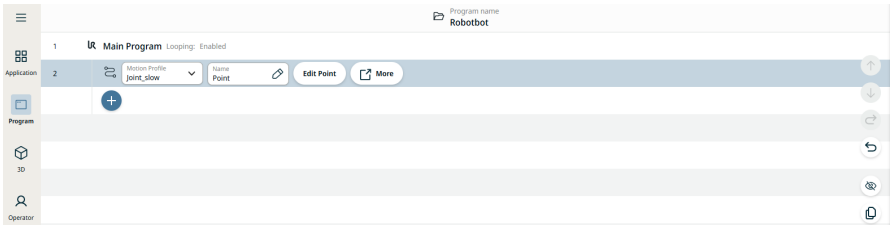
To access Joint Move command

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



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

- Motion Profile
- Name
- Edit Point

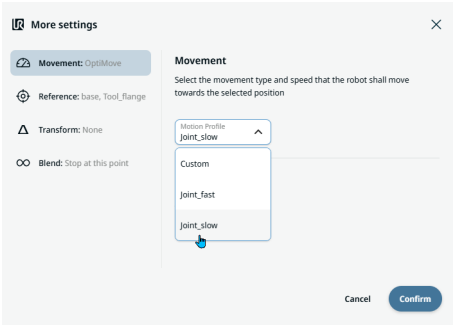


3. Tap the Motion Profile field and select the pertinent from the three choices are available:
    - Custom
    - Joint_fast
    - Joint_slow
4. Tap the Name field, and three tabs appear:
    - Value
    - Variable
    - Expression
5. Enter the data in the **Value**, **Variable**, and **Expression** tabs, and tap **Confirm**. See Value-Variable-Expression Tabs.
6. Tap **Edit Point**. A new screen appears with the robot arm interface.
7. Follow the steps in the section “To use Edit Point field” in the **Move to** command. See Move To command.
8. Tap **More** button.

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

Movement setting

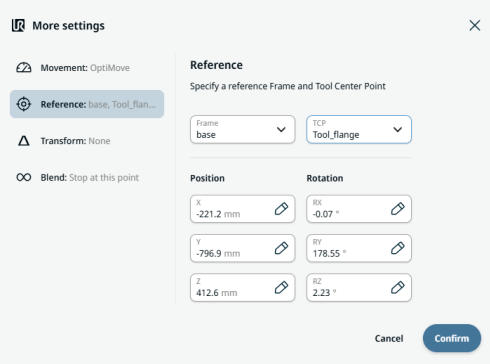
1. Tap the **Movement** setting on the left side. A **Motion Profile** field with the following choices appears on the right side. Joint_slow is the default.
   - Custom
   - Joint_fast
   - Joint_slow



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

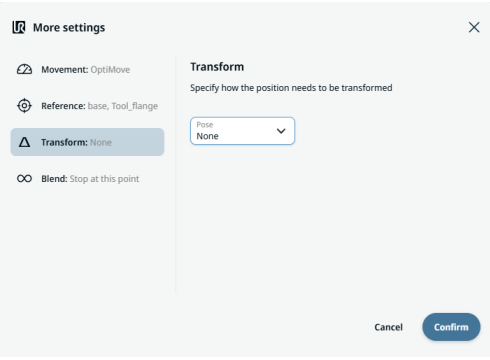
   In the TCP field, two choices are available:

   - 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

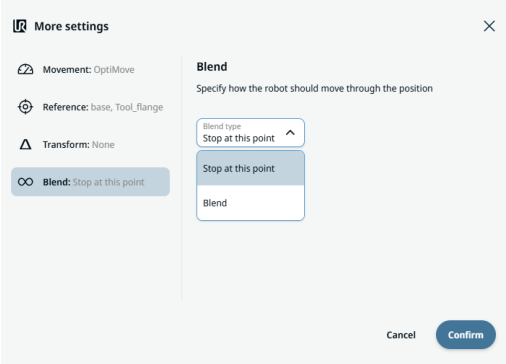
1. Tap the **Transform** setting.



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.

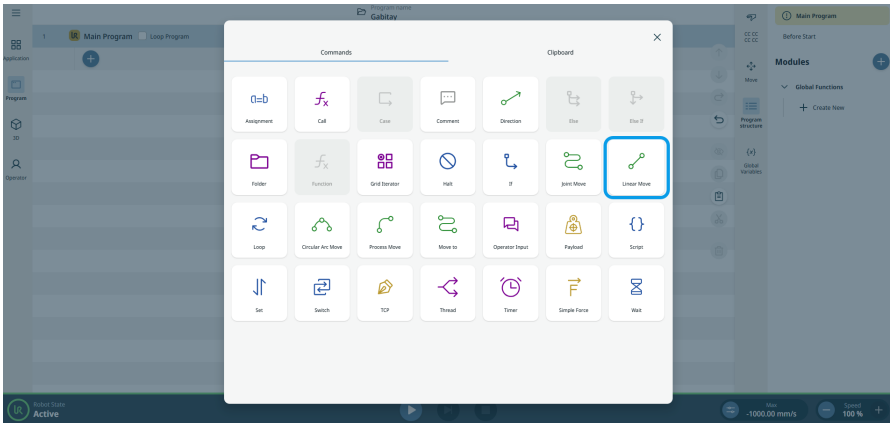
13.3.15. Linear Move

Description

The **Linear Move** command behaves in a similar way to the existing Move To node but supports Motions Profiles. This command 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.

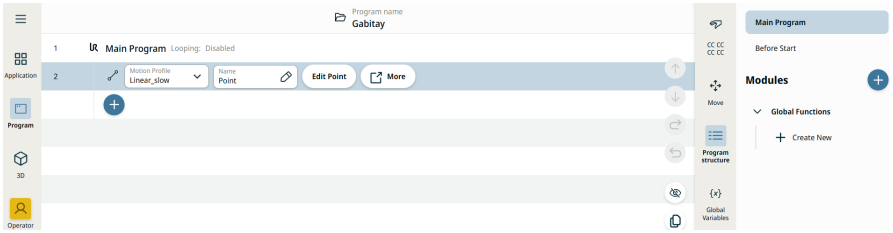
To access Linear Move command

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



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

- Motion Profile
- Name
- Edit Point

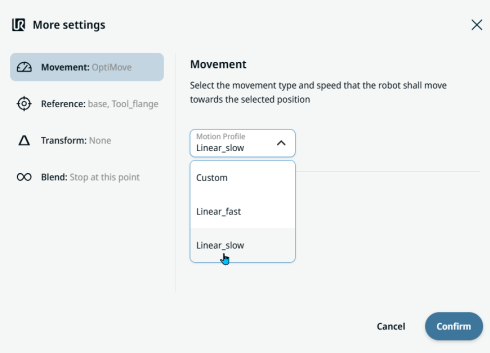


3. Tap the **Motion Profile** field and select the pertinent from the three choices are available:
   - Custom
   - Linear_fast
   - Linear_slow

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. Follow the steps in the section “To use Edit Point field” in the **Move to** command. See Move To command.
8. Tap **More** button. The **More settings** screen appears with four settings:
    - Movement
    - Reference
    - Transform
    - Blend

Movement setting

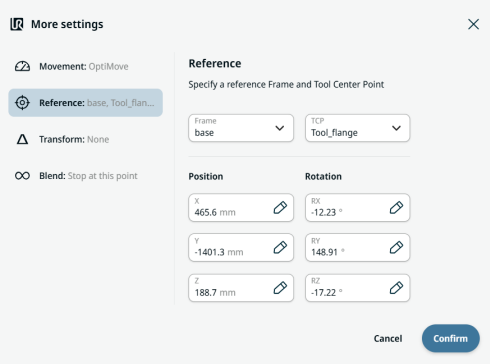
1. Tap the **Movement** setting on the left side. A **Motion Profile** field with the following choices appears on the right side. **Linear_slow** is the default.
    - Custom
    - Linear_fast
    - Linear_slow



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

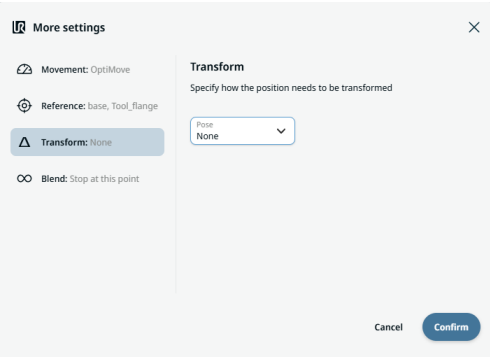
1. Tap the **Reference** setting.



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

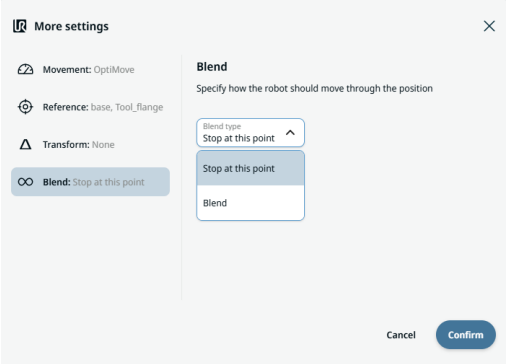
1. Tap the **Transform** setting.



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.

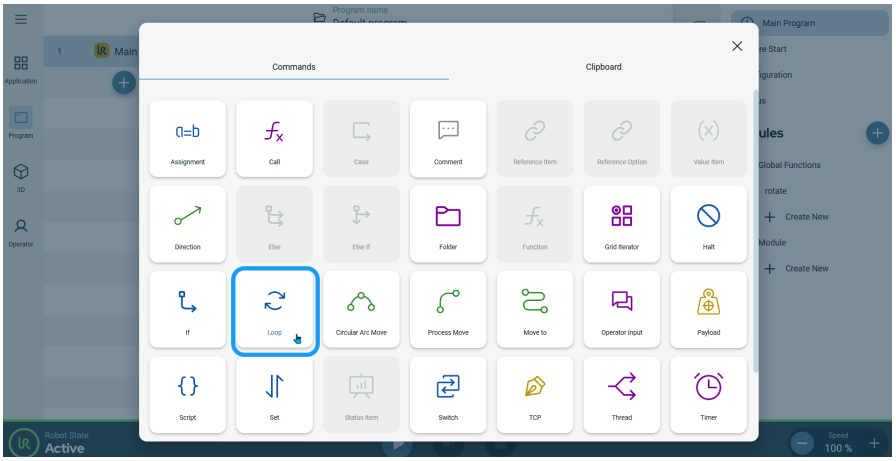
13.3.16. Loop

Description

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

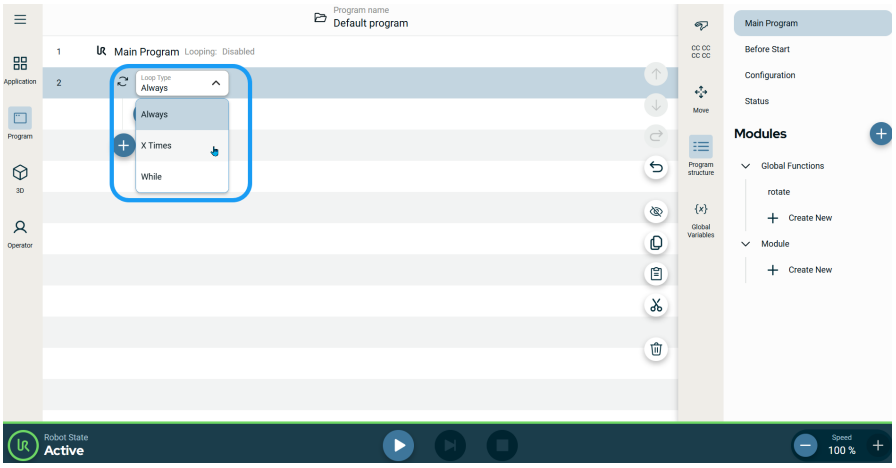
To access Loop command

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**.

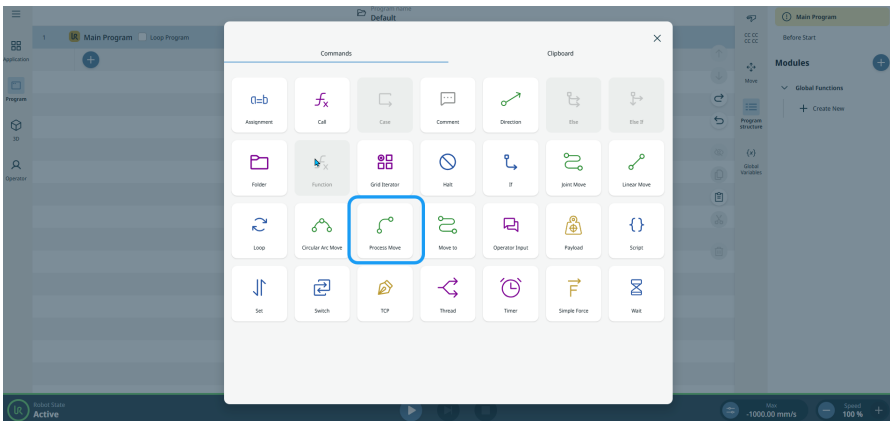
13.3.17. Process Move

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.



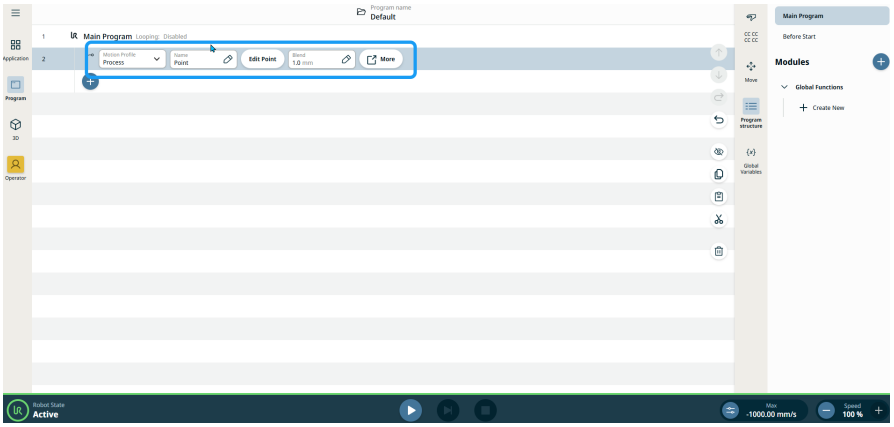
A node is inserted into the program tree. The node has three editable fields and two buttons.

These are the three fields:

- Motion Profile
- Name
- Blend

These are the two buttons:

- Edit Point
- More



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

   Custom motion profile enables you to define movement data in **Move to** node.

   Process motion profile is a predefined motion in **Move to** node.
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. (SeeMove To command.)
7. Tap the **Blend** field.

   Three tabs appear:

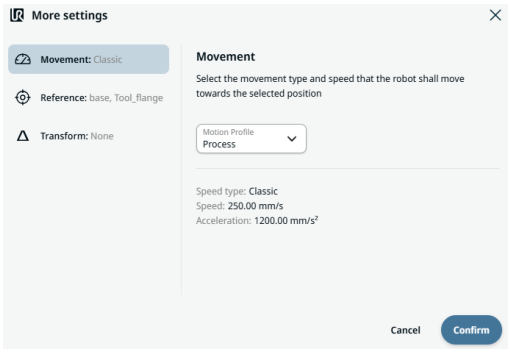
   - Value
   - Variable
   - Expression
8. Enter data in the **Value**, **Variable**, and **Expression** tabs, and tap **Confirm**. See Value-Variable-Expression Tabs.

To use More Option

1. Tap **More** button.

   More settings screen appears with three settings:

   - Movement
   - Reference
   - Transform



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

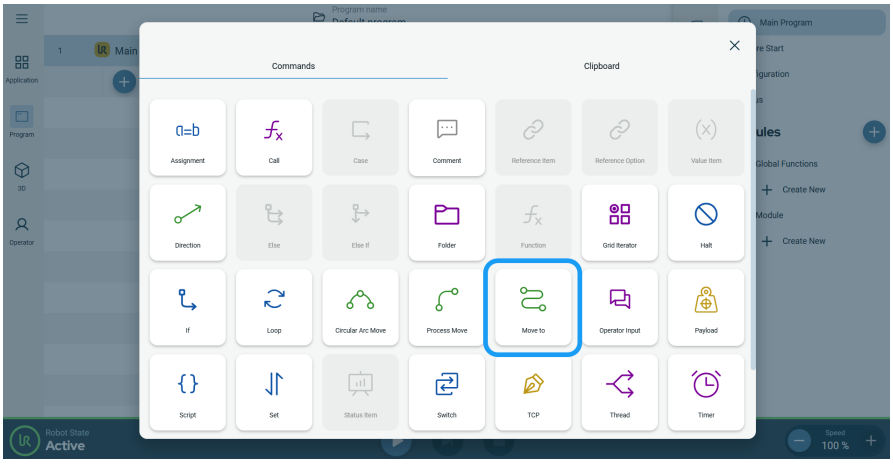
   A **Motion Profile** field with selections **Custom** and **Process** is on the right side.
3. If Custom is selected, a **Speed** field and **Acceleration** field appears.
4. Tap the **Speed** field, and the **Value**, **Variable**, and **Expression** tabs pop up.
5. Enter data in each tab, and tap **Confirm**. 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**.

13.3.18. Move To

Description The **Move to** command allows the robot to move from different points to a given location.

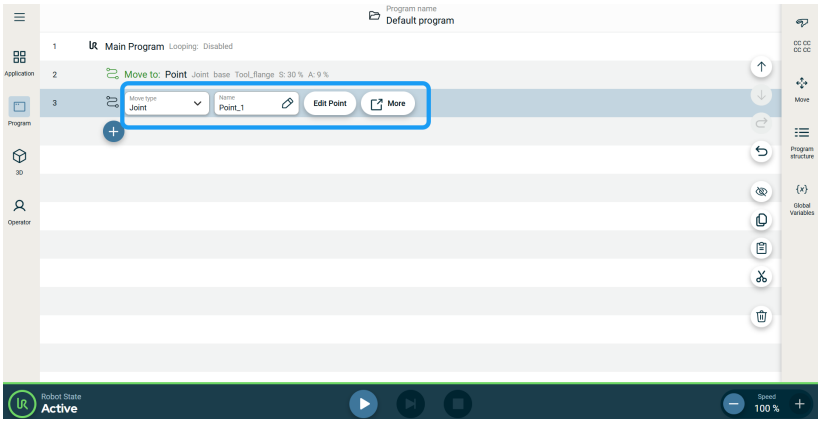
To access Move to command

1. Go to the main program screen. See Command Nodes.
2. Tap the **Move to** 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:

- Move type
- Name
- Edit Point



3. Tap the **Move type** field and choose either **Joint** or **Linear**.

   Joint. The robot makes movements that are calculated in the arm joint space.

   The joints are controlled to finish their movements at the same time, resulting in a curved path for the tool.

   Linear. The robot moves the tool center point (TCP) in linear path between waypoints.

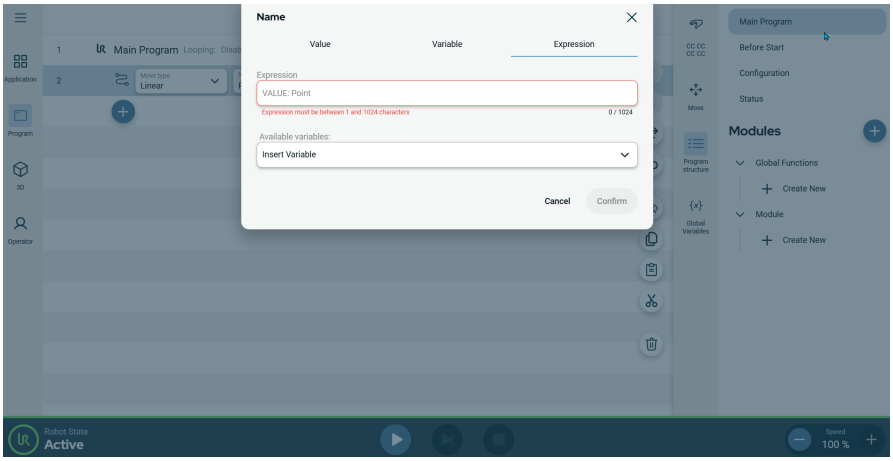
   Each joint performs a more complicated motion to keep the tool on a straight line path.

4. Tap the **Name** field.

   Three tabs appear:

   - Value
   - Variable
   - Expression

   See Value-Variable-Expression Tabs



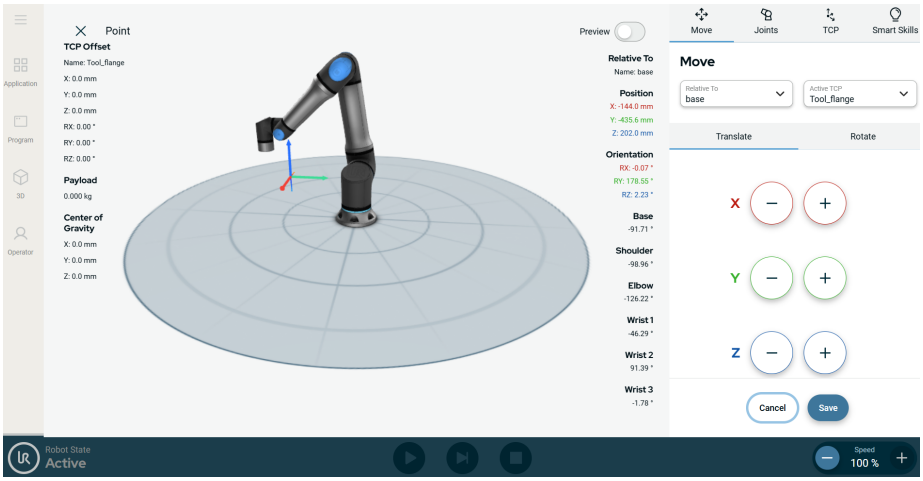
5. Input the data in the **Value** and **Expression** tabs.
6. Tap the **Variable** tab and the **Available** variables in the **Expression field**.
7. Choose **grid_iterator**.
8. Tap **Confirm**.

   Note: When **grid_iterator** is chosen as a variable, the Edit Point tab disappears.

To use Edit Point field

1. Tap the **Edit Point** field.

   A new screen appears with the robot arm graphical interface screen.

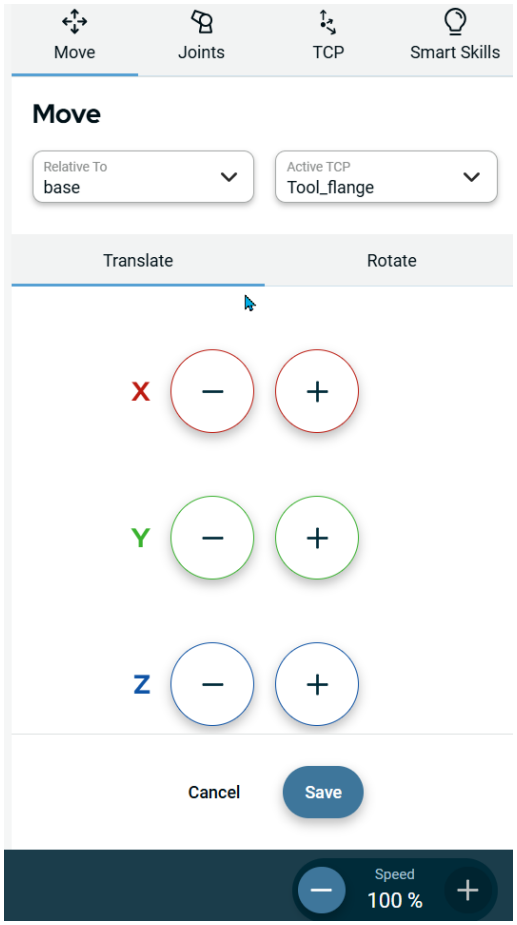


   On the right side of the screen, you can see four related tabs:

   - Move
   - Joints
   - TCP
   - Smart Skills

To use Move

1. Tap the **Move** tab.



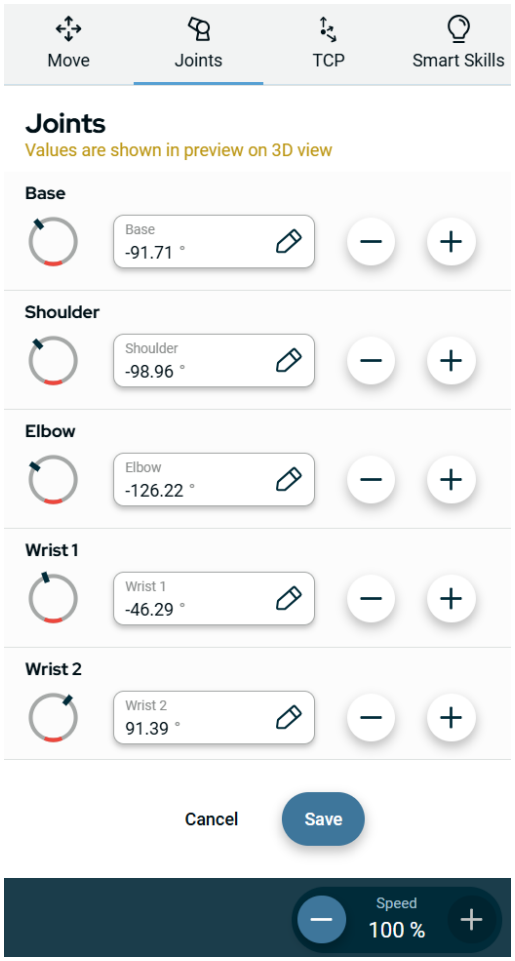
2. Tap the **Relative To** field to choose the robot position/part.

   Changes in coordinates are seen in the robot arm graphical interface screen.

3. Use the plus-minus button in the **Translate** and **Rotate** fields to reach your desired position or movement.

To use Joints

1. Tap **Joints** tab.



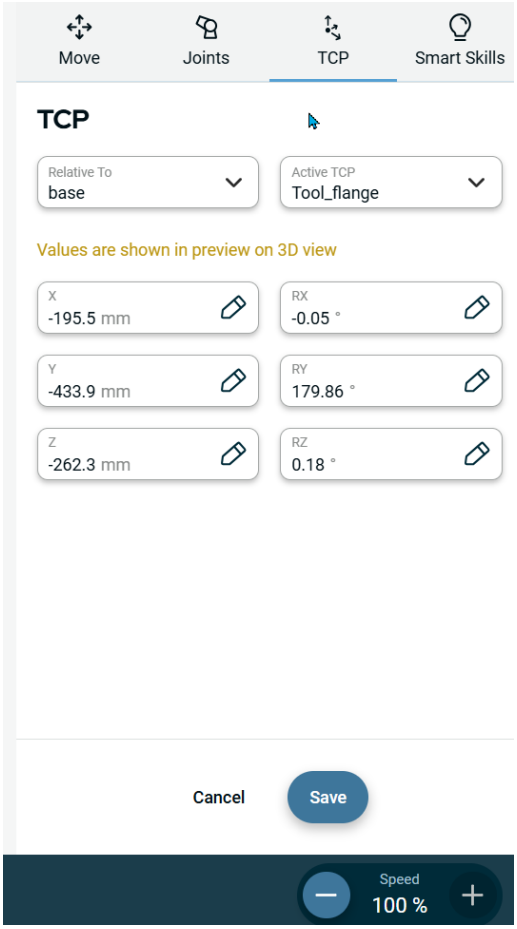
You can see the three options to input the values for the six parts of the robot arm:

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

2. Rotate clockwise or counterclockwise the control ring of each part to reach your joint value.
3. Tap the rectangular bar for each robot arm part.
4. Type and enter the value to edit and tap **Save**.
5. You can also use the plus-minus button to reach the joint value.

To use TCP

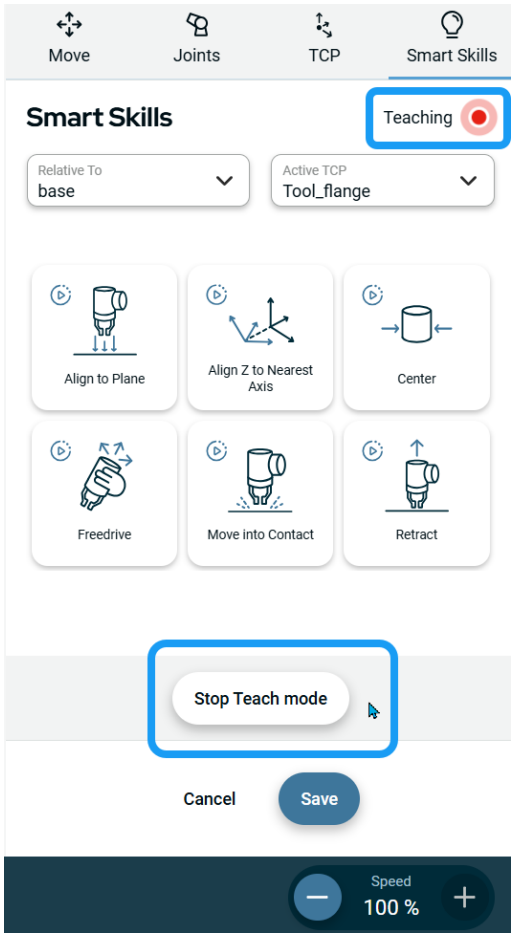
1. Tap **TCP** tab.



2. Inside the **Relative to** field, choose your relative point from the three options.
3. Tap the **Active TCP** field.
4. Enter the value in mm in the fields of **X**, **Y**, **Z** and in degrees in **RX**, **RY**, **RZ** fields.
5. Tap **Confirm**.

To use Smart Skills

1. Tap the **Smart Skills** tab.



Six smart skills functionalities are given for you to move the robot:

- Align to plane
- Align Z to nearest axis
- Center
- Freedrive
- Move into contact
- Retract

2. Tap the **Teach mode** button below the Smart Skills icons.

   A blinking red light at top right turns on, signifying **Teaching**.

3. Select a smart skill.

   A corresponding movement is shown on the robot arm screen.

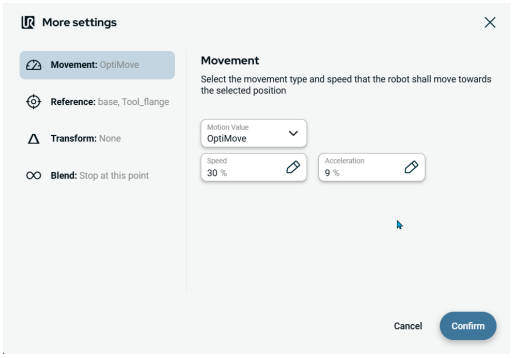
4. Tap **Stop Teach** mode.

5. To exit the robot arm interface screen, tap **X** above TCP Offset located in the main screen, or tap **Cancel** found at the right side of the screen.

To use More option

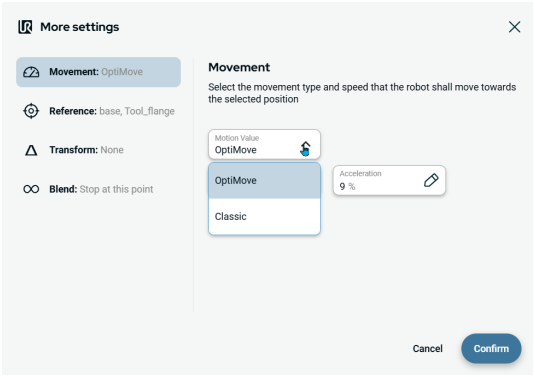
1. Tap the **More** button.

   More settings screen appears.



2. Four more settings appear:
   - Movement
   - Reference
   - Transform
   - Blend

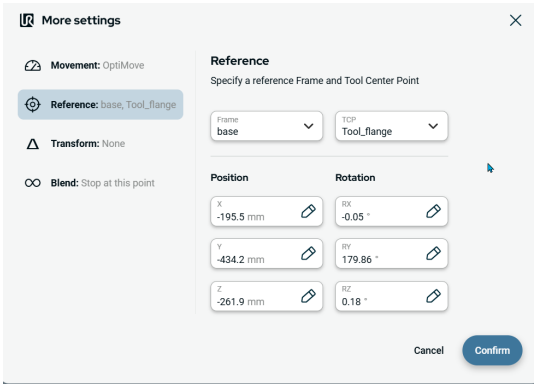
To use Movement setting



1. Tap **Movement** on the left side of the screen.
2. Tap the **Motion Value** field.
3. Choose either **OptiMove**or **Classic**.
4. On the **Speed** tab, enter the percentage of your preferred speed and tap **Confirm**
5. On the **Acceleration** tab, enter the percentage of your preferred acceleration and tap **Confirm**.

   See OptiMove to use its setting.

To use Reference setting



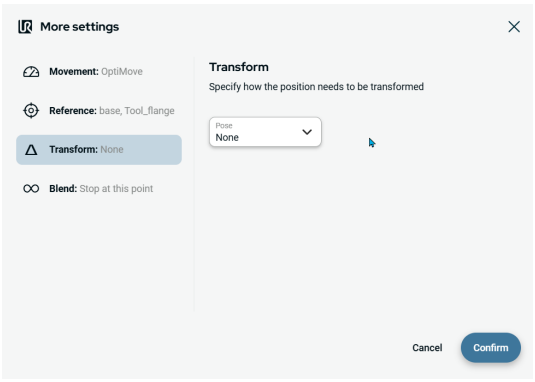
1. Tap **Reference** on the left side of the screen.

   Reference is the frame which the node is relative to.
2. Tap the **Frame** field.

   Four choices are available:

   - world
   - base
   - flange
   - tcp
3. Choose your preferred frame.
4. On the **TCP** field to the right, choose either **Active_TCP** or **Tool_flange**.
5. In the **Position** column, tap the **X**, **Y**, **Z** tabs.
6. Enter your preferred value in mm for each tab and **Confirm**
7. In the **Rotation** column, tap the **RX**, **RY**, **RZ** tabs.
8. Enter your preferred value in degrees for each tab and **Confirm**

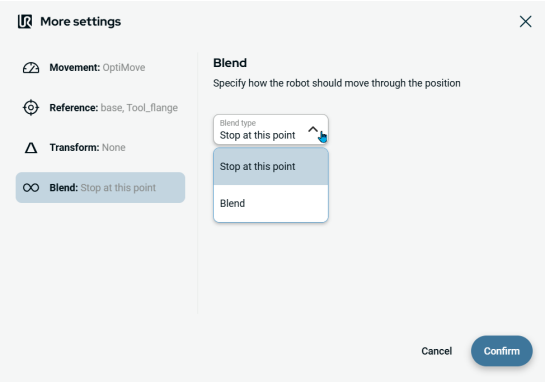
To use Transform setting



1. Tap **Transform** on the left side of the screen.

   Transform allows you to specify a pose offset to the **Move to** node.
2. Tap the **Pose** field.
3. Choose your preferred pose and tap **Confirm**.

To use Blend setting



1. Tap **Blend** on the left side of the screen.

   Blending enables the robot to smoothly transition between two trajectories, without stopping at the waypoint between them.
2. Tap the **Blend type** field.
3. Choose either **Stop at this point** or **Blend**.

   If **Blend** is selected, you can specify a blend radius for how smoothly to transition in the **Radius** field.
4. Input the value, variable, and expression of the radius and tap **Confirm**.

   See Value-Variable-Expression Tabs.

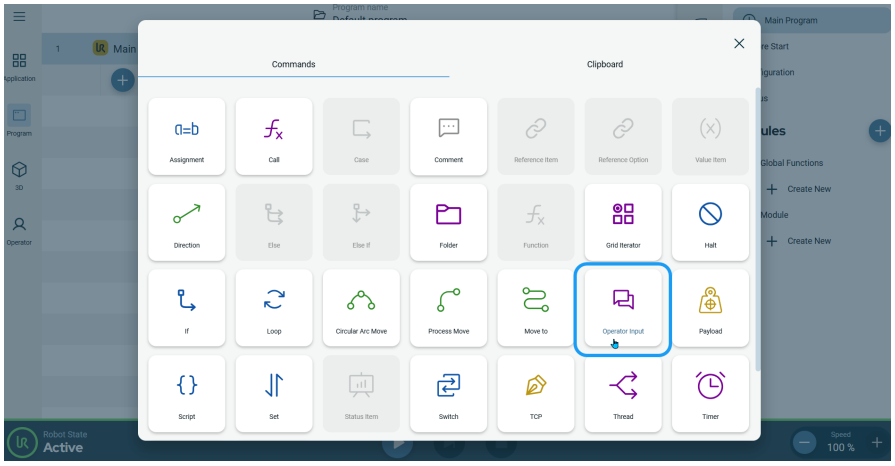
13.3.19. Operator Input

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:

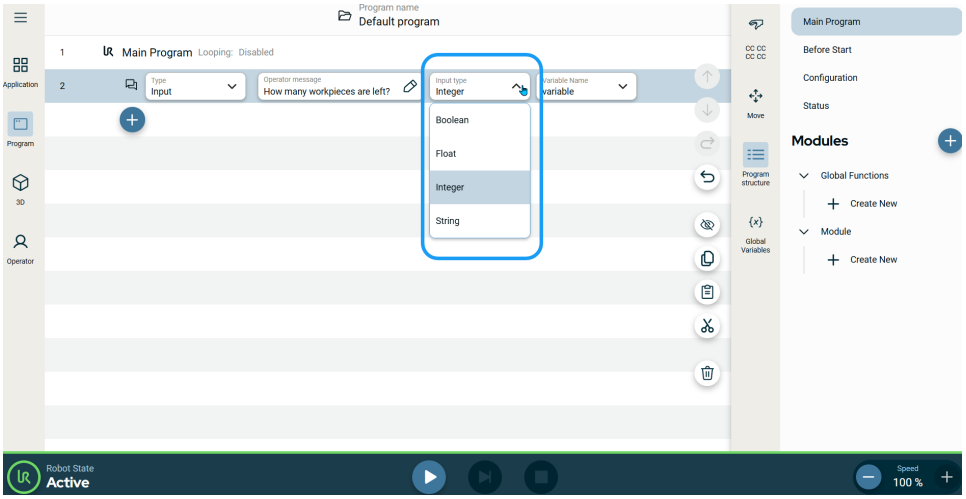
   - Type
   - Operator message
   - Input type
   - Variable Name

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**.

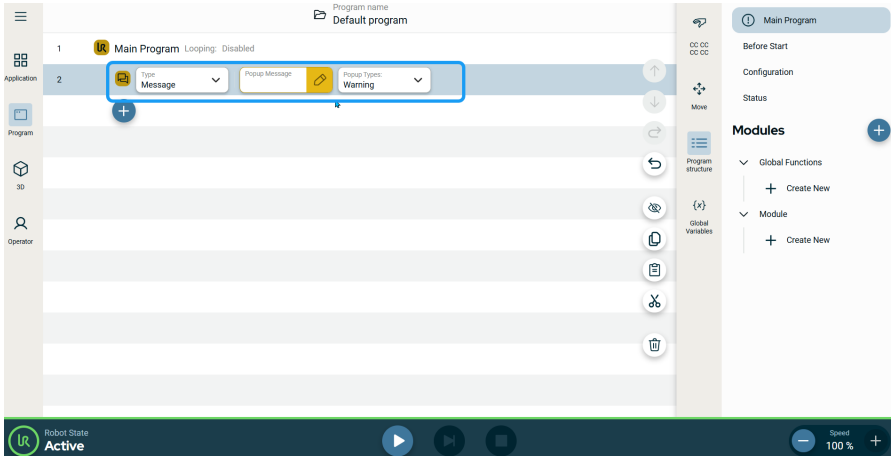
   The program stops at this point, displays the message from the Message field, and waits for a response.

   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



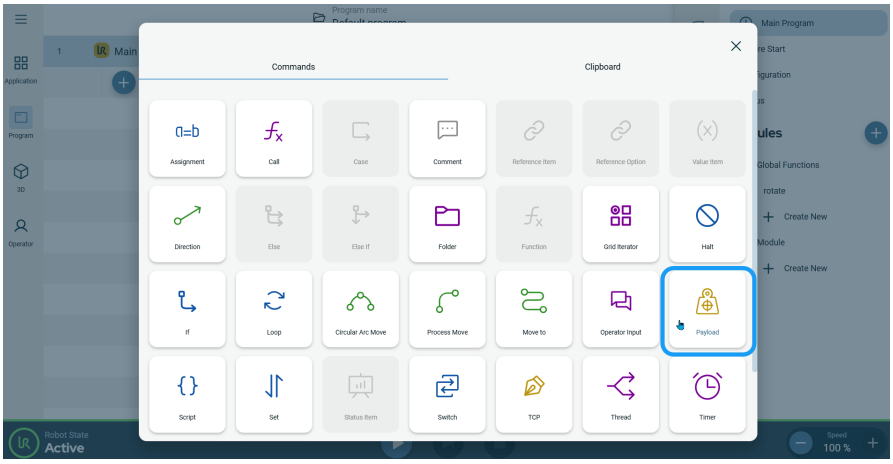
10. Tap **Popup Message** field.
11. Create your message and tap **Confirm**.
12. Tap **Popup Types** field.
13. Select the type from the three choices:
    - Error
    - Warning
    - Info

13.3.20. 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:

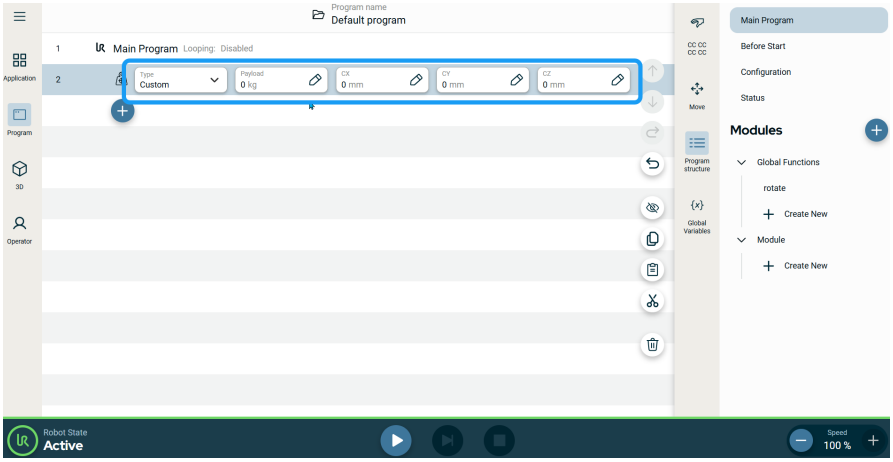
- Custom
- Robot

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

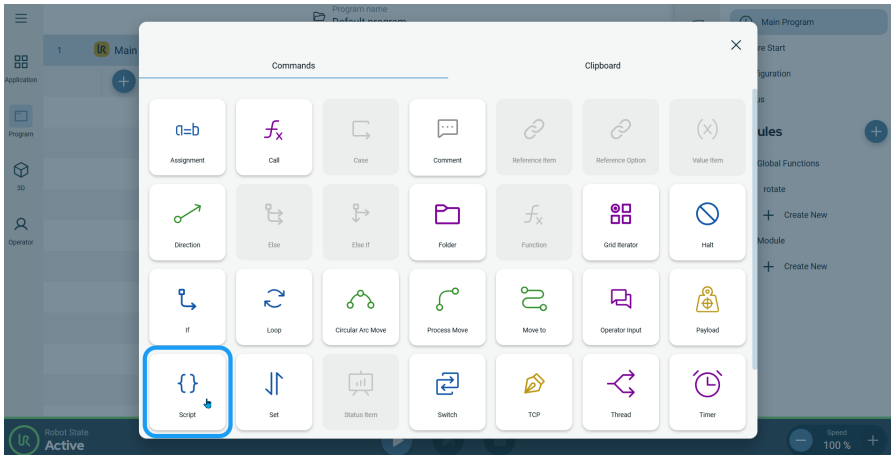
   Note: The Variable tab is inactive.
5. Make the input in each tab and tap **Confirm**. See Value-Variable-Expression Tabs.
6. Tap the **CX** field and repeat step 5.
7. Tap the **CY** field and repeat step 5.
8. Tap the **CZ** field and repeat step 5.

13.3.21. Script

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

To access Script command

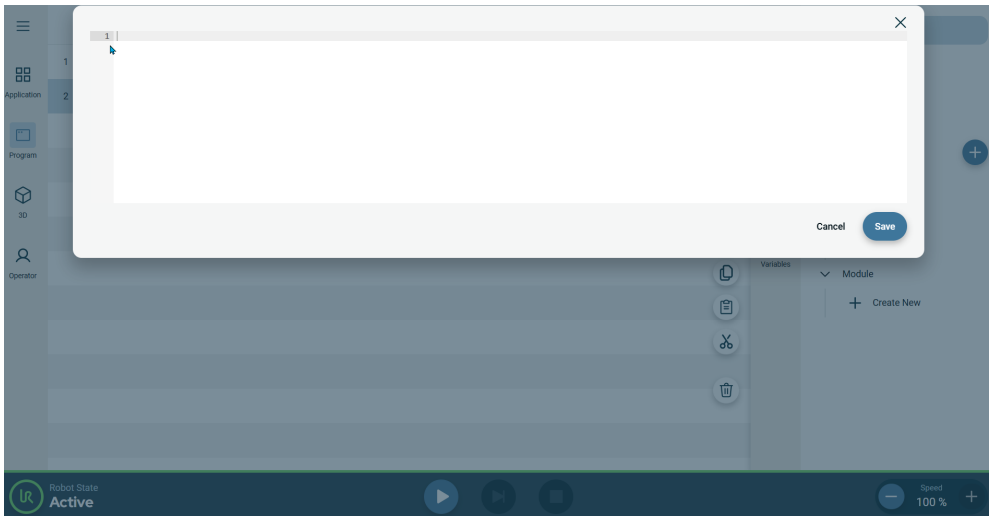
1. Go to the main program screen. See Command Nodes.
2. Tap the **Script** icon in the Commands toolbox.



A node is inserted into the program tree. The node has an editable **Name** field and a **Set Script** button.

3. Tap the **Name** field.
4. Enter and edit your script name, and tap **Confirm**.
5. Tap the **Set Script** button.

   A new screen appears.



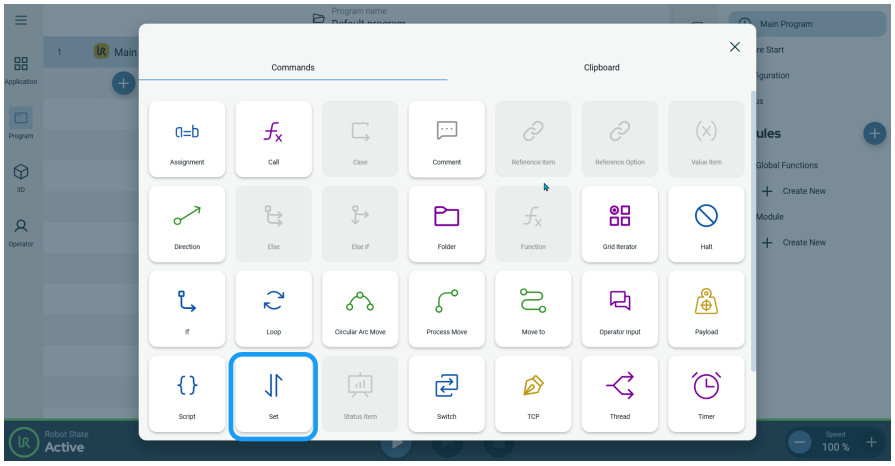
6. Enter the script and tap **Save**.

13.3.22. Set

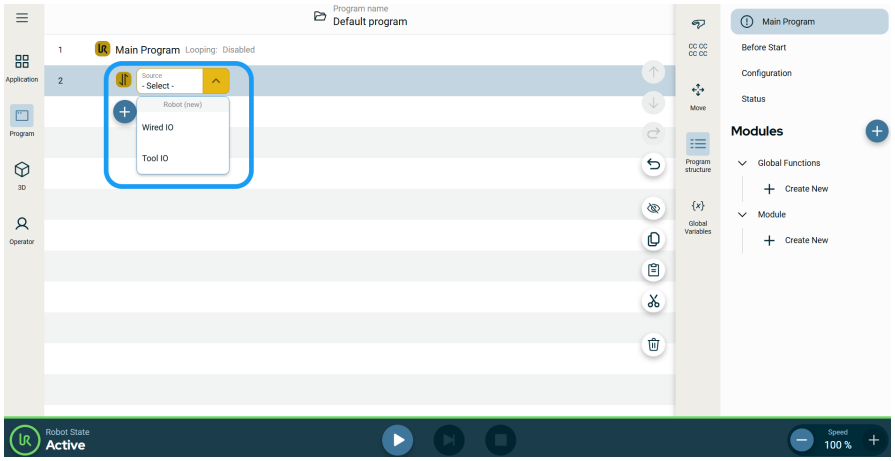
Description The **Set** command enables you to set a given value to either digital or analog outputs.

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.

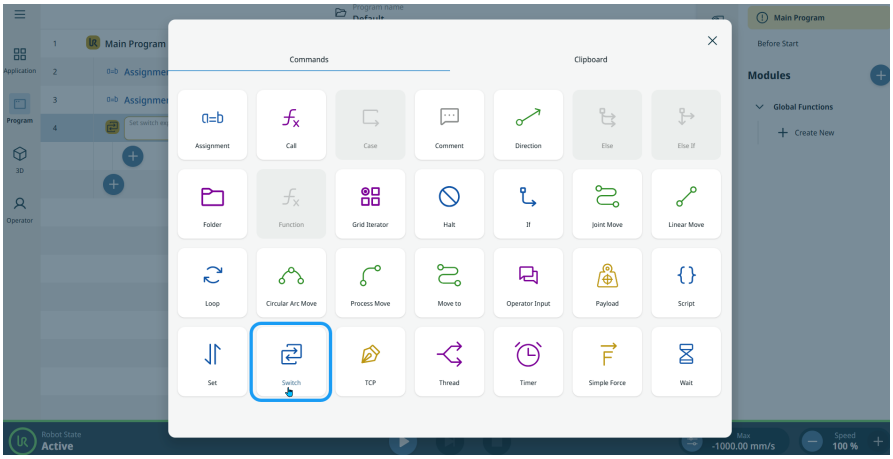
    A **Value** field is inserted to the right with three options:
    - Use Variable
    - High
    - Low
13. Tap **Use Variable**, and the **Variable** and **Expression** tabs appear.
14. Enter data in the two tabs, and tap **Confirm**. See Value-Variable-Expression Tabs.
15. You can also choose either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

13.3.23. 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.

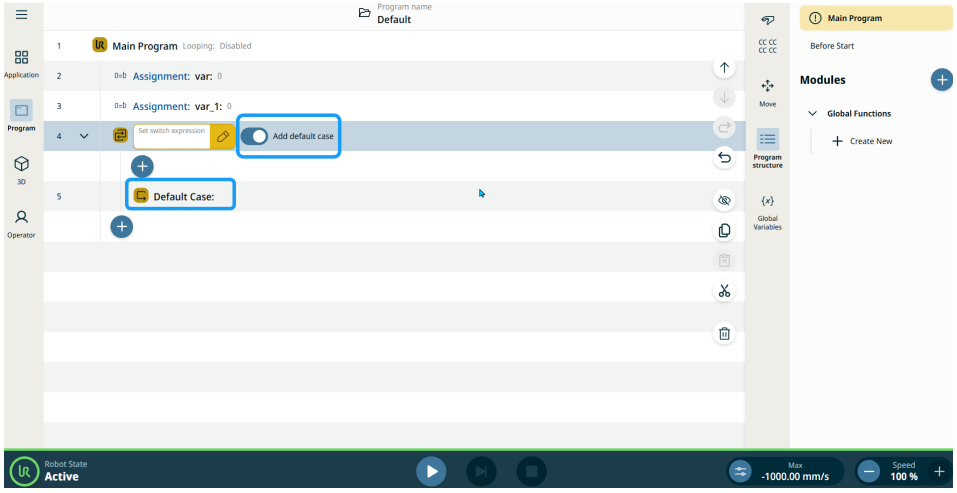


A node with **Set switch expression** field and **Add default case** button is inserted into the program tree.

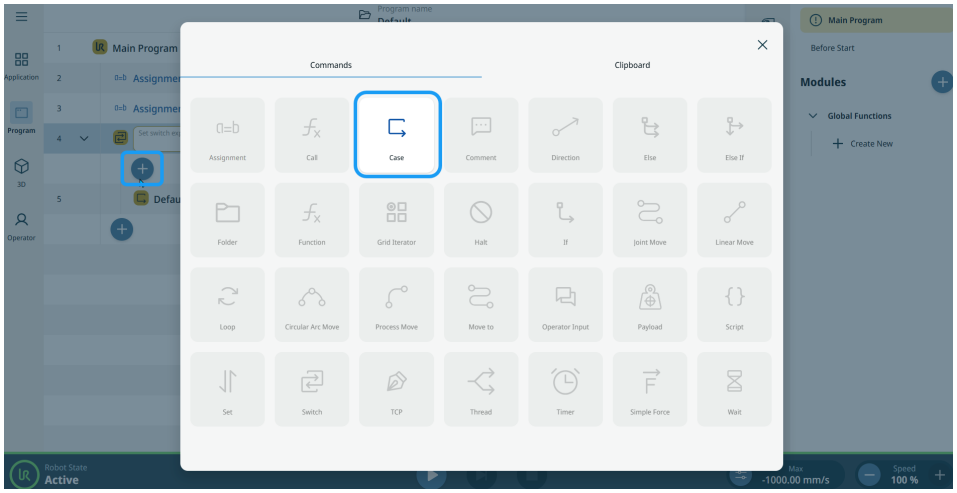
3. Tap the **Set switch expression** field, and two options are available:
   - Variable
   - Expression
4. Tap **Variable** and select or switch based on an existing variable or an expression.

   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.

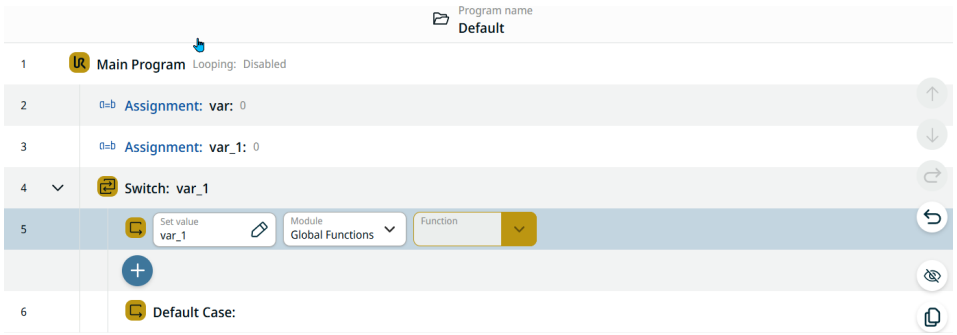


7. Tap the **add icon** inside the Switch node to add a case node.
8. Tap the **Case** icon in the Commands toolbox. See Case.



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.



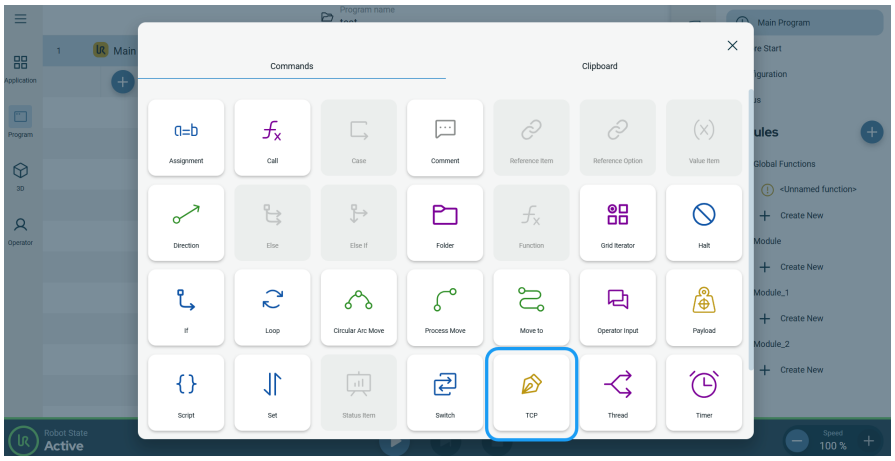
13.3.24. TCP

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.

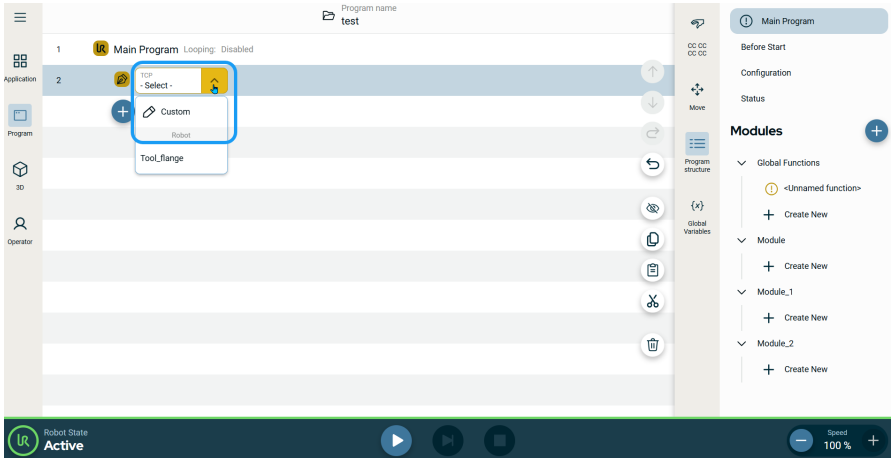
To access TCP command

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.

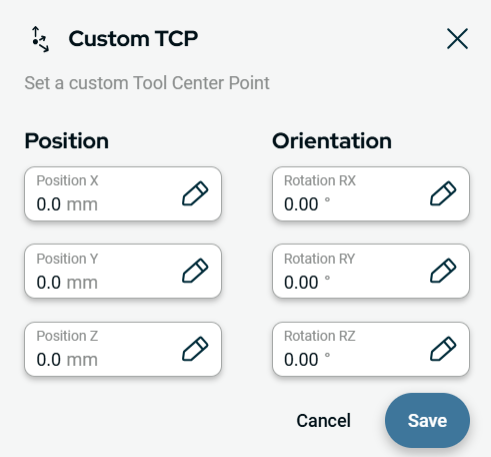


4. Select **Custom**.

   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**.

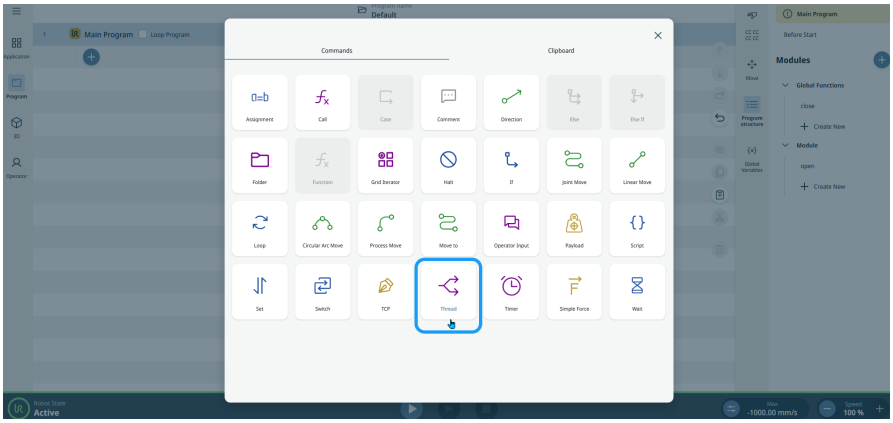
13.3.25. Thread

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.

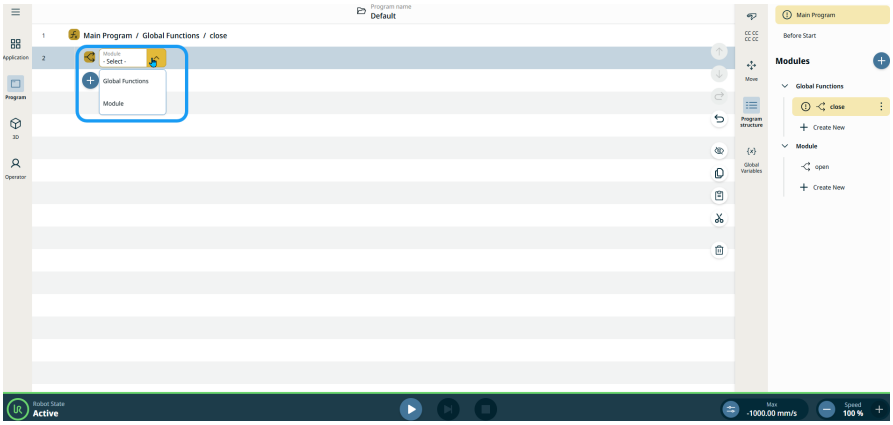
To access Thread command

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



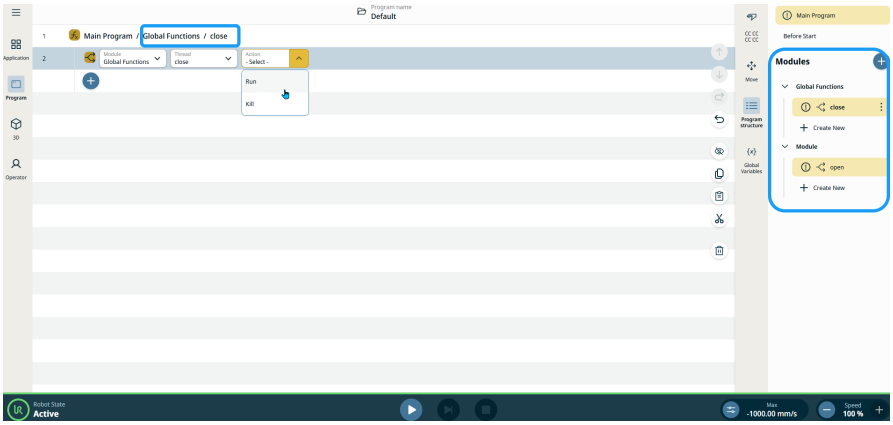
A node with a **Module** field is inserted into the program tree.

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

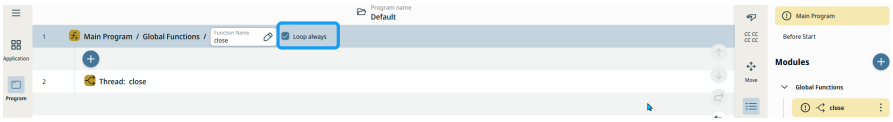


4. Select either **Global Functions** or **Module**, and a **Thread** field appears at the right.

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.

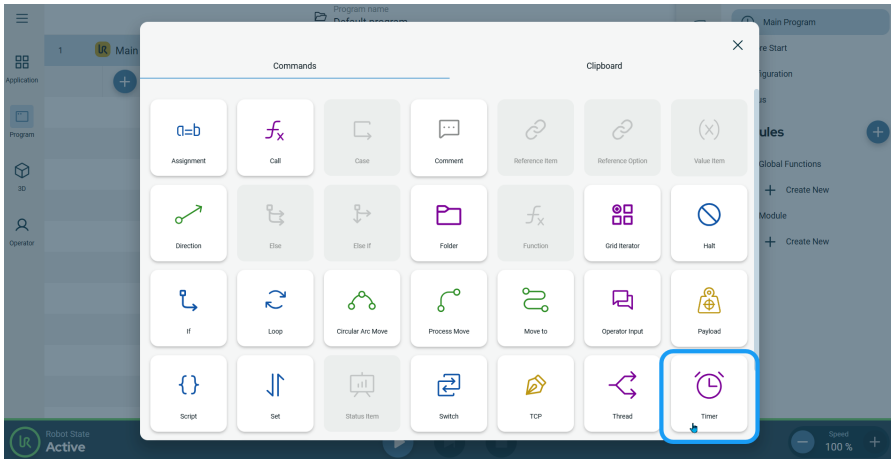
   Threads that are meant to run for the entire duration of the program execution can be started from **Before Start** section of the **Main Program**.

13.3.26. 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.

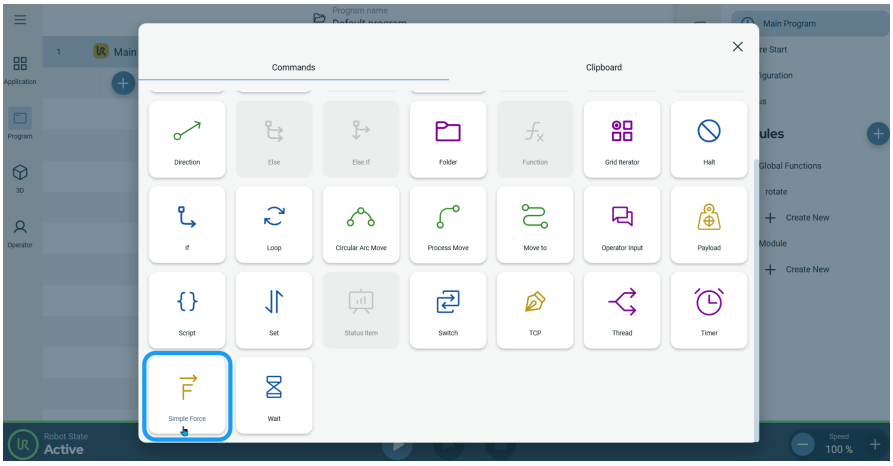
13.3.27. Simple Force

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

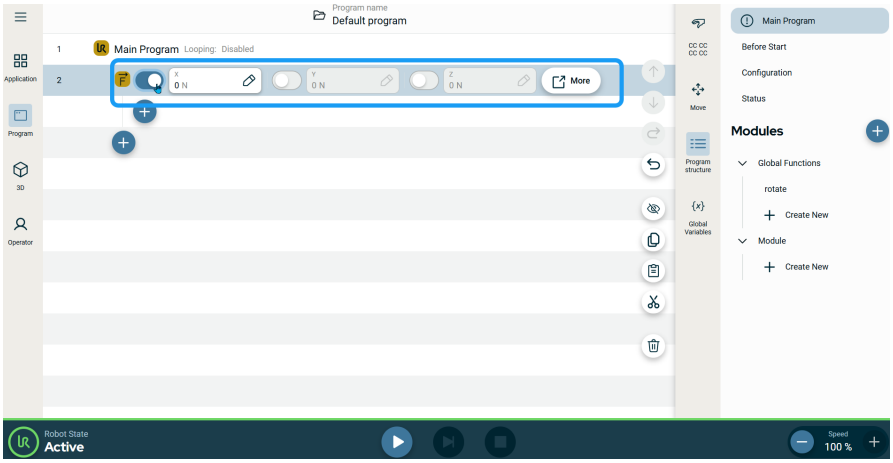
1. Go to the main program screen. See Command Nodes.
2. Tap the **Simple Force** icon in the Commands toolbox.



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.

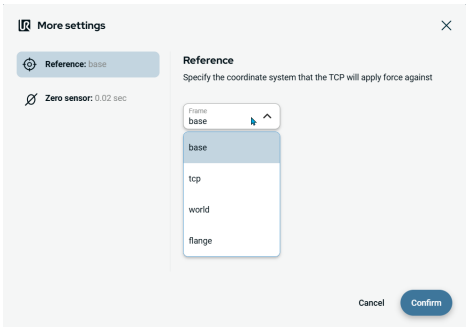
   A new screen appears containing two advanced settings:

   - Reference
   - Zero sensor

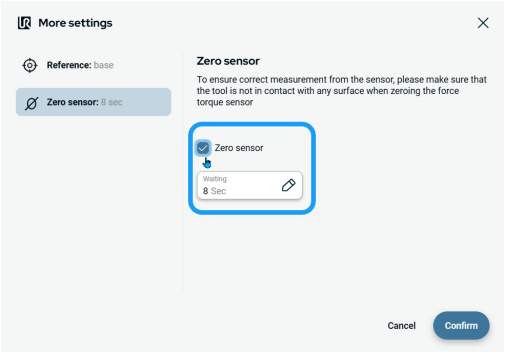
2. Tap **Reference** on the left side.

   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**.

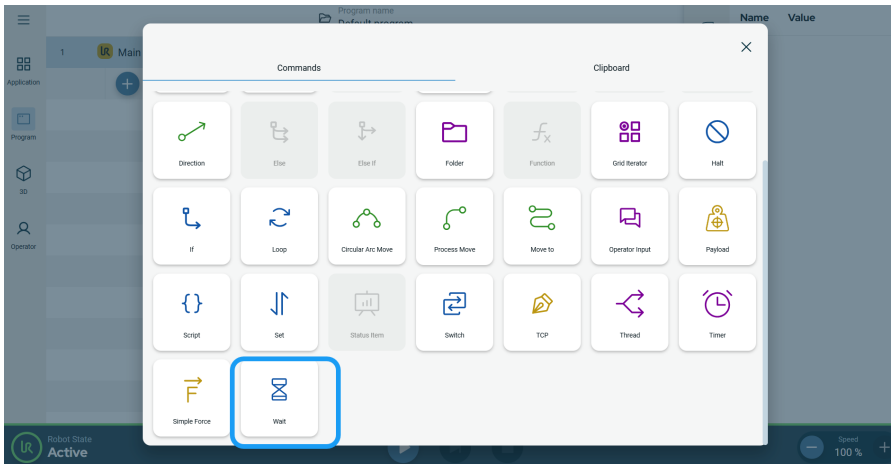
13.3.28. Wait

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.



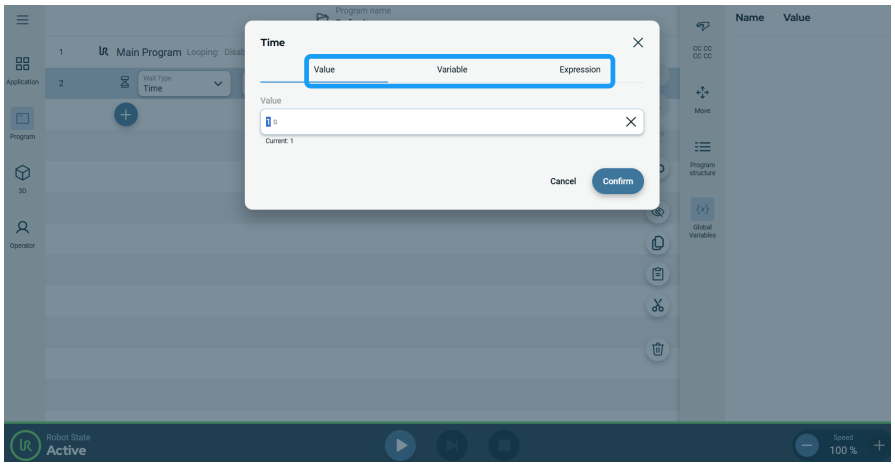
A node with two editable fields is inserted into the program tree:

- 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.

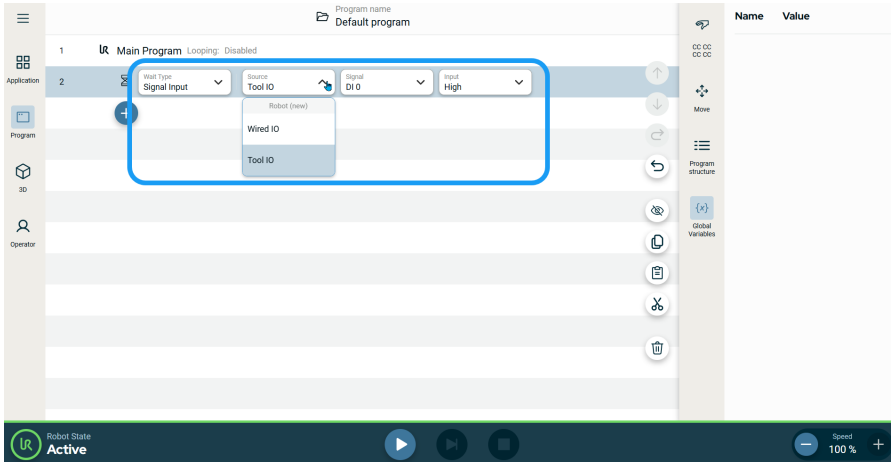
4. On the **Time** field, enter the **Value** and **Expression**, and tap **Confirm**. See Value-Variable-Expression Tabs.



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**.

14. The First Program

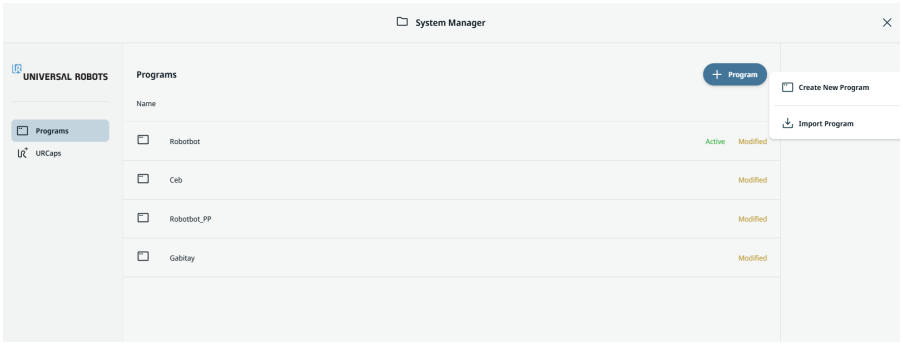
Description

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.

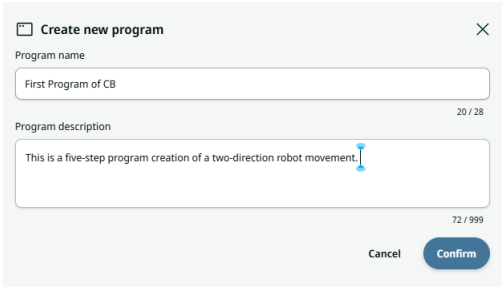
To create a simple program

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

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



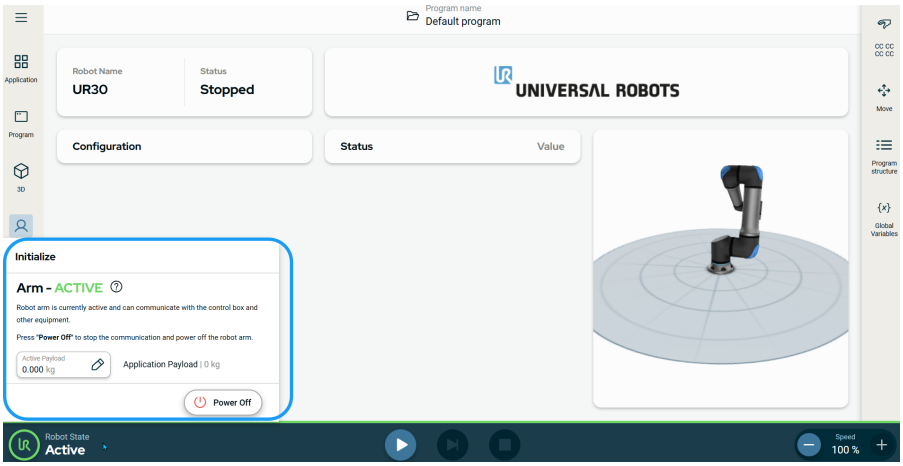
3. The **Create new program** box appears, and in it you enter the **program name** and **description** in the two fields provided. Tap **Confirm**.



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.

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

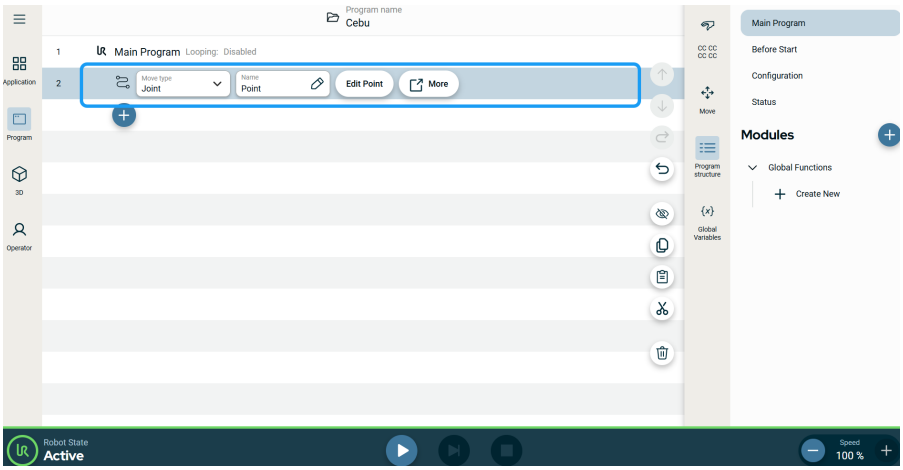
5. Tap **Power On**, then **Unlock**.



In the footer, you see the robot state as **Active**, and the activated button/slider of **Play**, **High Speed Manual**, and **Speed**.

6. On the main navigation, tap the **Program** menu icon. See Command Nodes.
7. Tap Move To command.

   Note: In PolyScope X, the Waypoint node is merged to the **Move to** command.

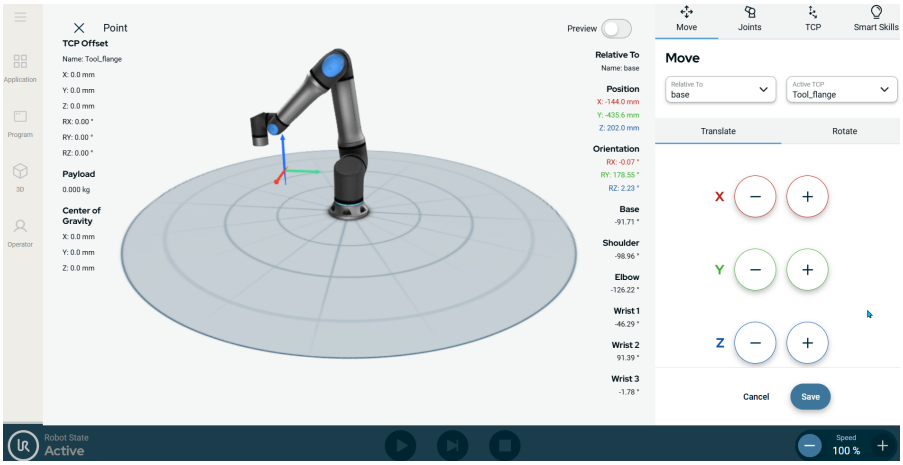


Two editable fields and two tabs are inserted into the program tree:

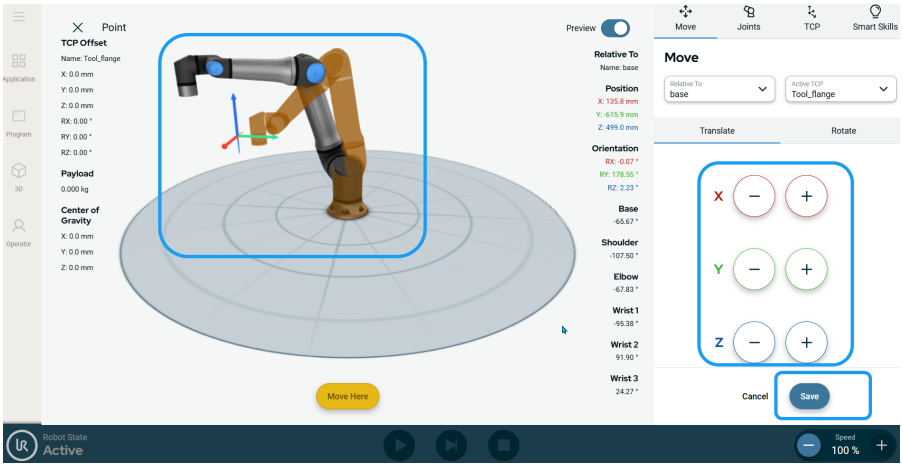
- Move type
- Name
- Edit Point
- More

8. Tap the **Edit Point** tab.

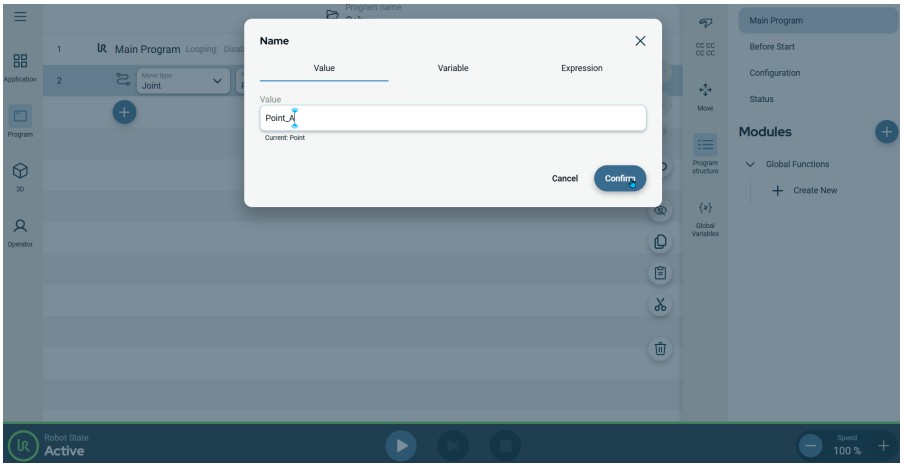
   The screen with the robot arm in 3D appears. To the right of the screen, you see the Move menu.



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

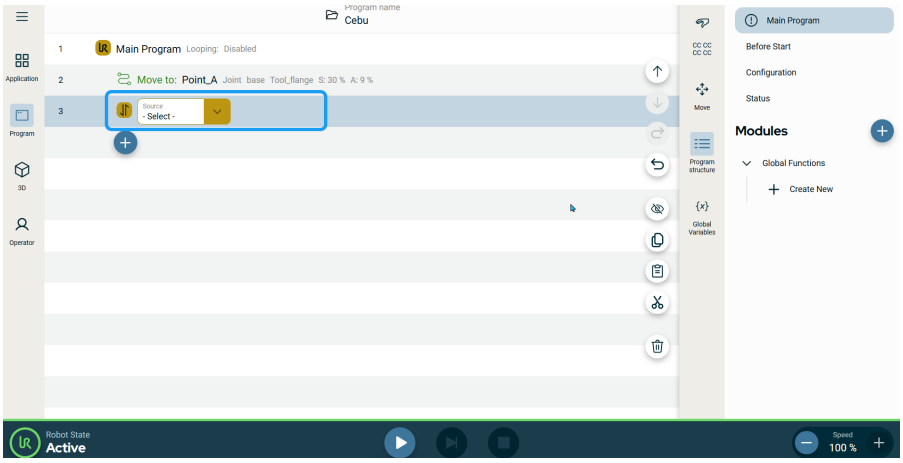


10. Tap the **Name** field.
11. In the **Value** editor tab, enter **Point_A** as the name and **Confirm**.

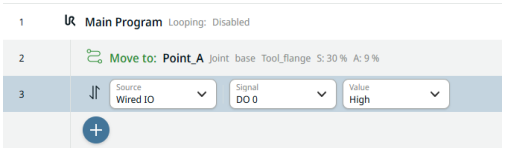


12. Tap the add icon in the program tree.
13. Tap the **Set** command icon. See Set.

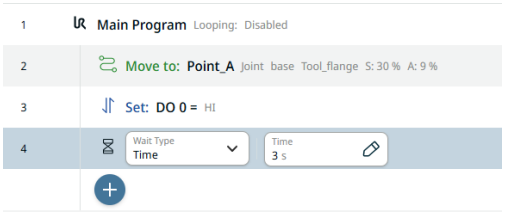
    The set node is inserted into the program tree below the **Move to** command.



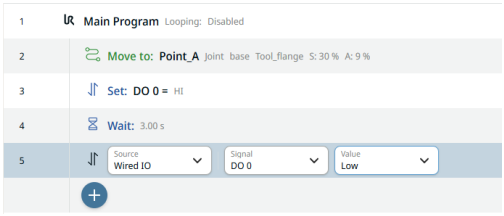
14. Tap the **Source** field and select **Wired IO**.
15. A **Signal** field is added. Select **DO 0**. A **Value** field is added to the right and select **High**.



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

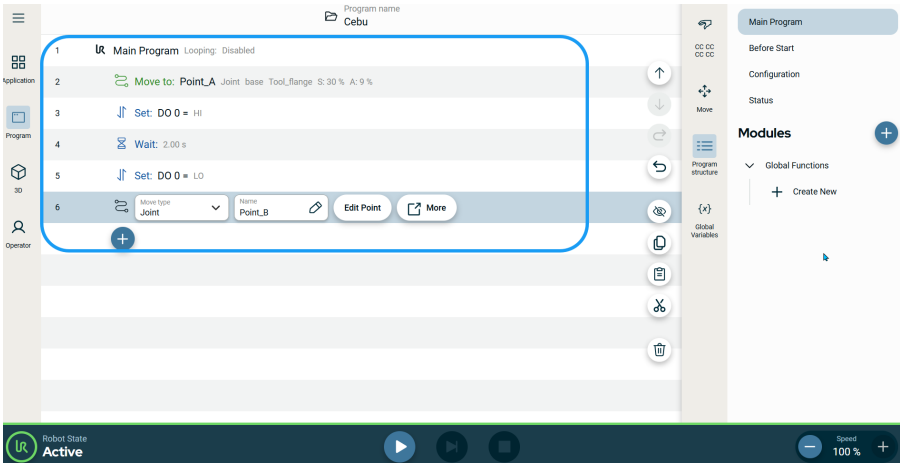


19. To create a second Set node, repeat **steps 12 to 14**.
20. In the **Signal** field, select **DO 0**. A Value field is added to the right and select **Low**.



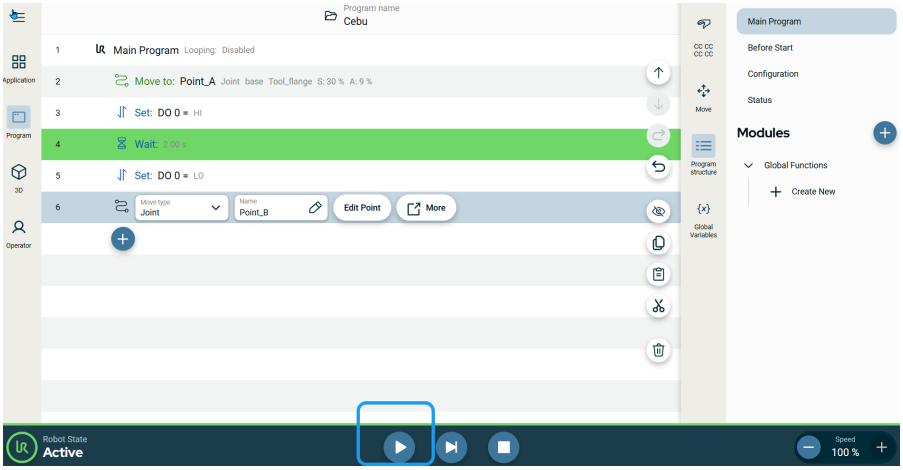
21. Tap the add icon.
22. Repeat **steps 7 to 10** to create a second waypoint. Move the robot arm position different to Point_A.
23. Enter **Point_B** in the **Value** tab and **Confirm**.

    The main screen shows you the first created program.

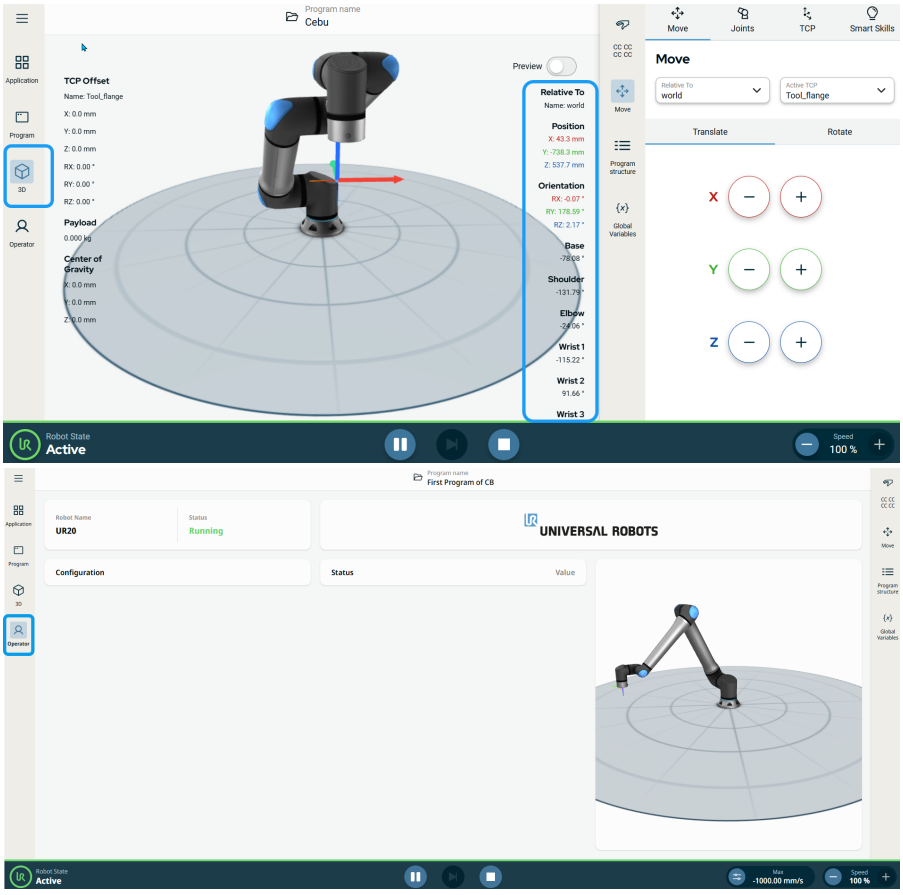


24. Tap the **3D Viewer** icon on the main navigation.
25. On the footer, tap the **Play** button.

    When you are in the Program menu, you see the program tree in green color is the current program execution.

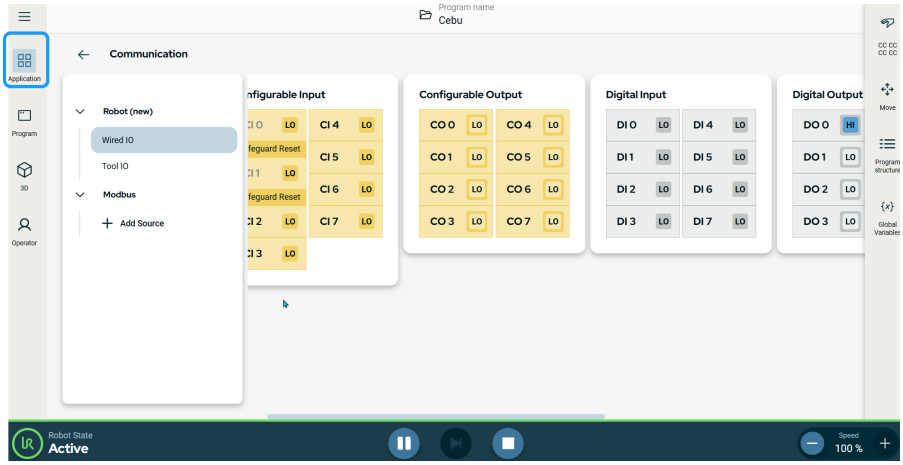


26. When you are in the 3D and Operator viewer, you see the robot move. You see the three-dimensional graphics and joint information update as the robot moves. You can tap again the **Play** button to see the robot move.



27. Go to the **Application** tab.
28. 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.



15. Application Tab

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

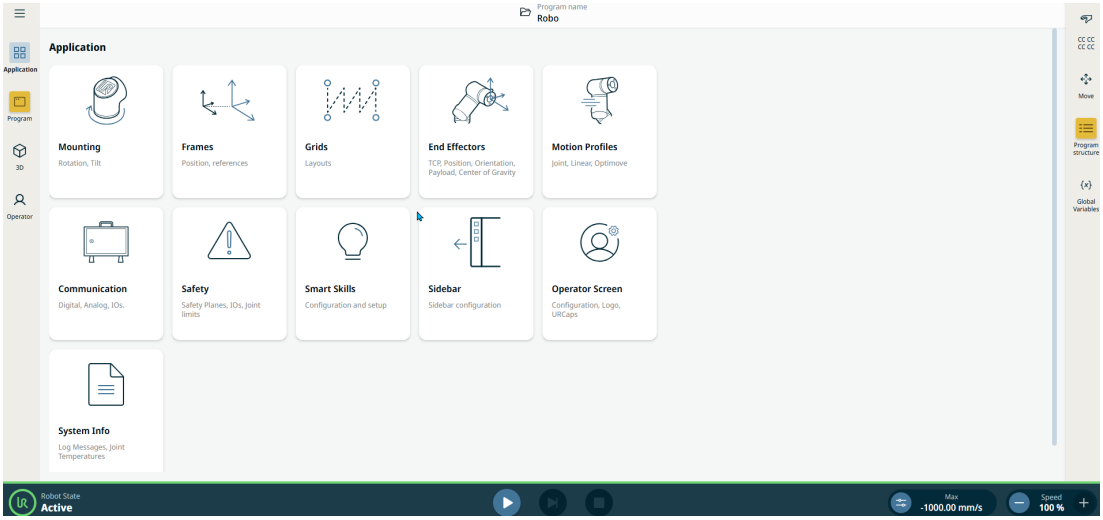


Figure 1.4: *Application screen displaying application buttons.*

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

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

15.1. Communication

Description

The Communication application allows you to monitor and set the live IO (input-output) signals from/to the robot control box.

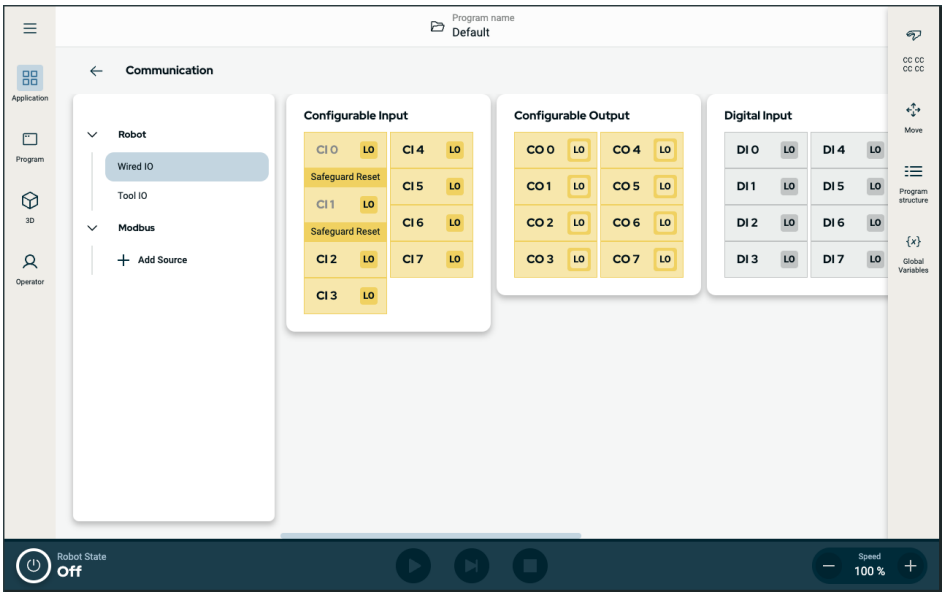
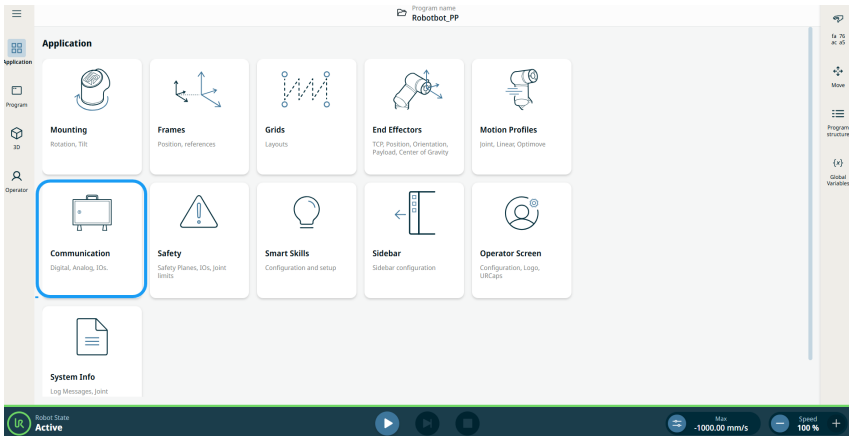


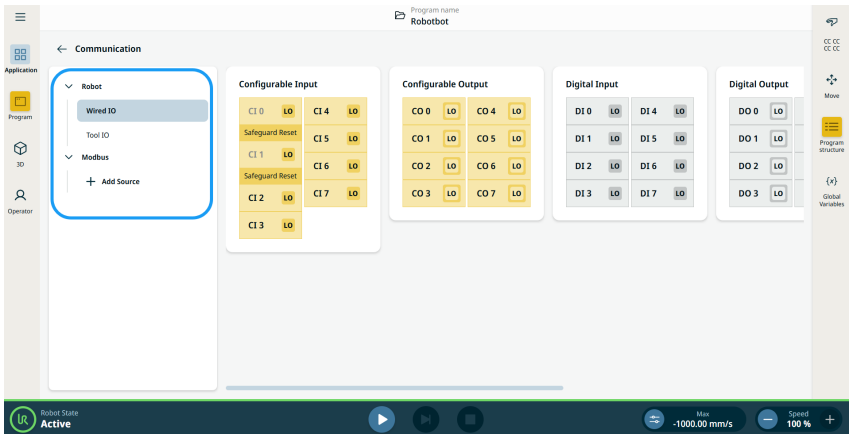
Figure 1.5: *Communication screen displaying IOs.*

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**

  Wired IO is used to monitor and set the live IO signals from/to the control box.
- **Robot Tool IO**
- **Modbus**

The right panels show the editable fields of the chosen function.

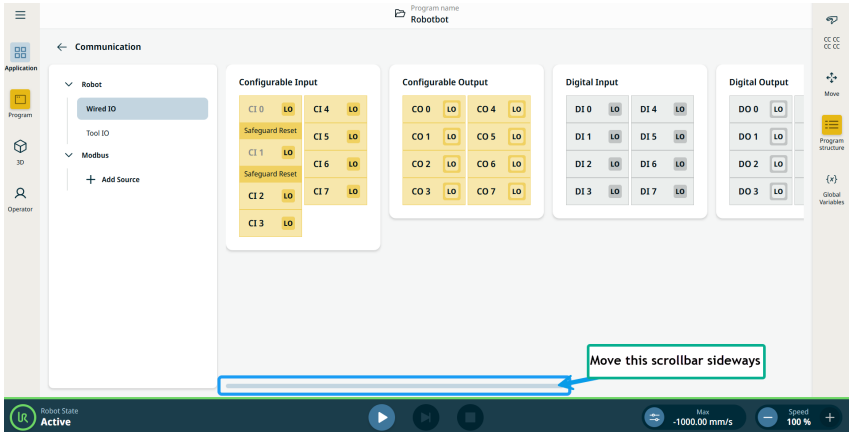
To access Wired IO

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

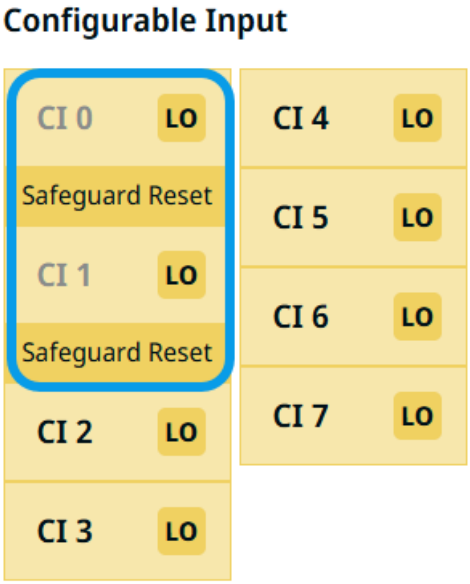


Input is used to start, stop, or pause the program, or as an external freedrive button.

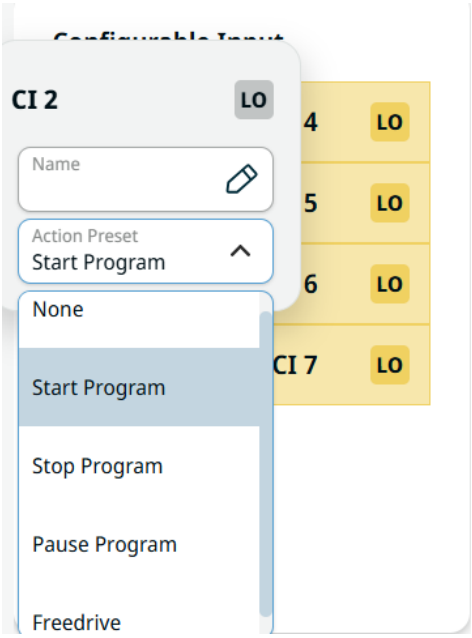
Output shows whether the robot is running or not.

Configurable Input

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.



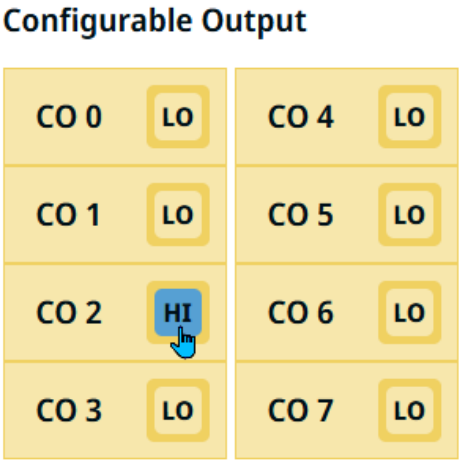
1. Tap **CI 2**, and the **Name** and **Action Preset** fields appear. In the **Name** field, create a name and **Confirm**. The new name can now be seen in the communication screen.



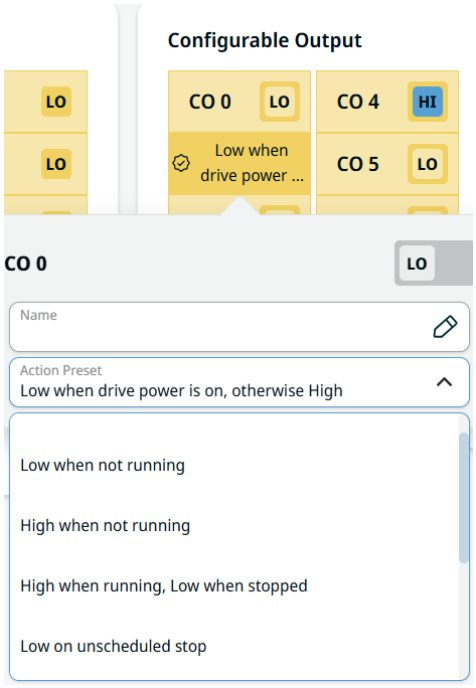
2. Tap the **Action Preset** field, and five choices are available:
   - **None**
   - **Start Program**. Starts the loaded program.
   - **Stop Program**. Stops the loaded program.
   - **Pause Program**. Pauses the loaded program.
   - **Freedrive**. Sets the robot in freedrive mode, just like the 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

The **Configurable Output** field ranges from CO 0 to CO 7.



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.



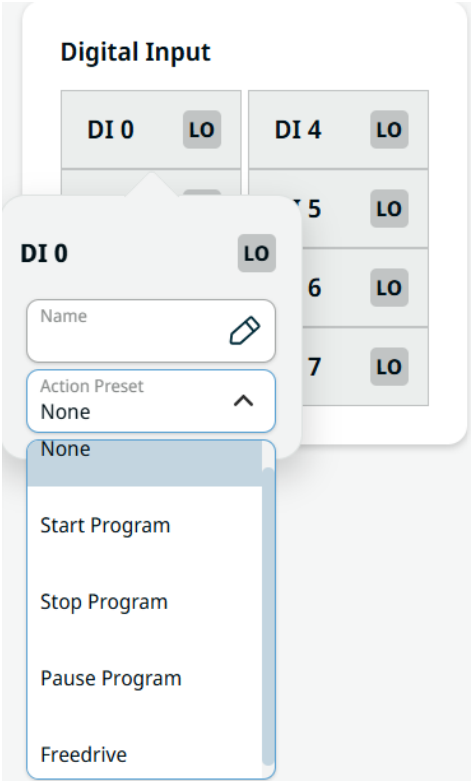
4. Tap the **Action Preset** field, and nine choices are available:
   - **None**
   - **Low when not running**. Output state is LO. Program state is stopped or paused.
   - **High when not running**. Output state is HI. Program state is stopped or paused.
   - **High when running, Low when stopped**. 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

The **Digital Input** field ranges from DI 0 to DI 7.

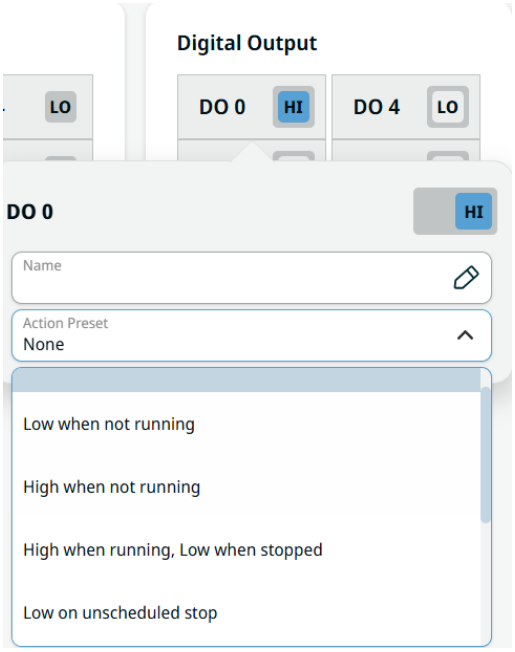
1. Tap **DI 0**, and the **Name** and **Action Preset** fields appear.
2. In the **Name** field, create a name and **Confirm**. The new name can now be seen in the communication screen.



3. Repeat steps 3 onward of Configurable Input for DI 1 to DI 7.

Digital Output

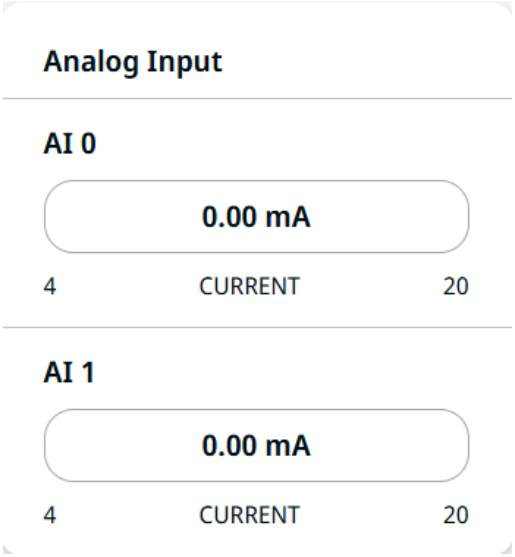
The **Digital Output** field ranges from DO 0 to DO 7 and are set independently to either high or low.



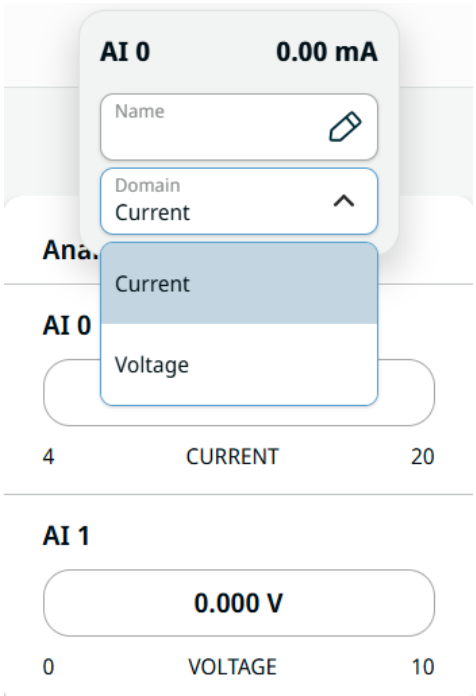
Follow all the steps as in Configurable Output.

Analog Input

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.



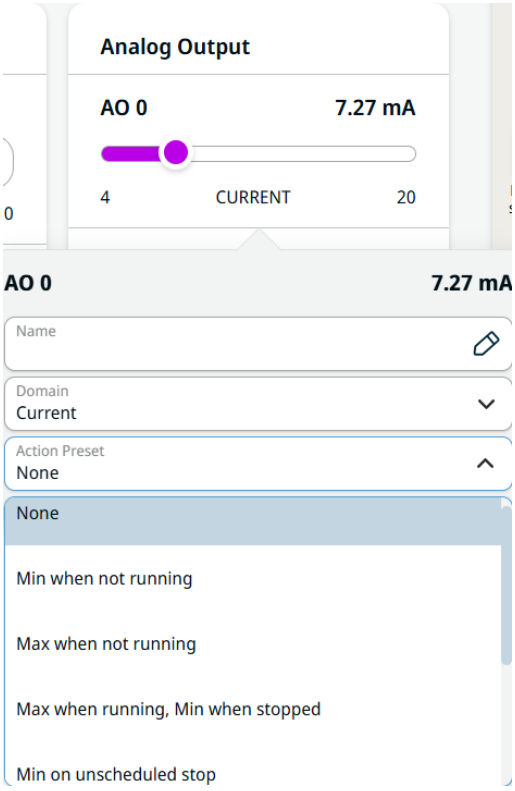
1. Tap **AI 0**. The **Name** and **Domain** fields appear.
2. In the **Name** field, create the name you prefer.



3. Tap the **Domain** field, and you can choose either Current or Voltage.
4. Tap **AI 1**.
5. Repeat steps 2-3.

Analog Output

Analog output has two fields for AO 0 and AO 1. As with analog input, it can be set to 4-20 mA.



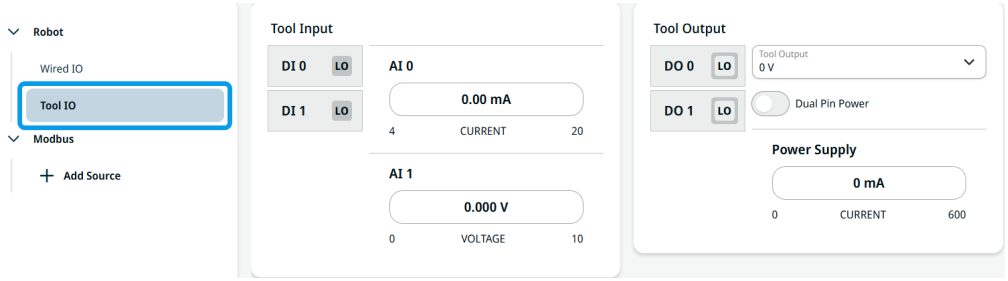
1. Tap the purple slider of each field to adjust the current or voltage settings.
2. Tap **AO 0**.
3. In the **Domain** field, you can choose either Current or Voltage.
4. Tap the **Action Preset** field, and eight choices are available:
   - None
   - Min when not running
   - Max when not running
   - 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.

To access Tool IO

Tool IO monitors and sets the live IO signals from/to the control box.

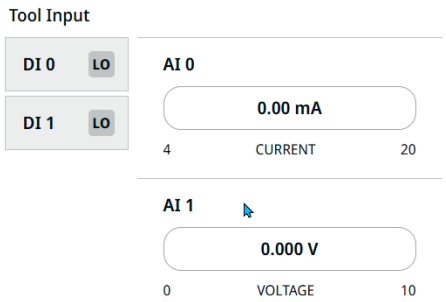
Tap **Tool IO** on the left panel. You can see the editable fields of Tool Input and Tool Output on the right.



Tool Input

The **Tool Input** is composed of four fields:

- DI 0
- DI 1
- AI 0
- AI 1

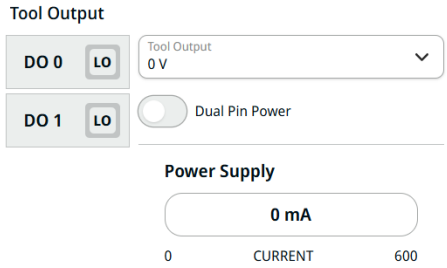


1. Tap **DI 0**, and the **Name** and **Action Preset** fields appear.
2. In the **Name** field, create a name and **Confirm**.
3. Tap the **Action Preset** field and select from the five available choices.
4. Repeat steps 1-3 for **DI 1**.
5. Tap **AI 0**. A **Name** and **Domain** fields appear.
6. 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

The **Tool Output** is composed of five fields:

- DO 0
- DO 1
- Tool Output
- Dual Pin Power
- Power Supply

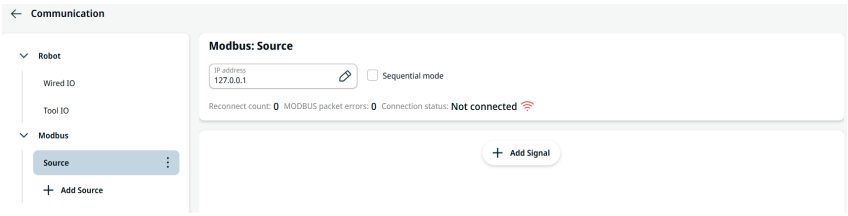


1. You can click the **square button** beside each DO to choose either LO or HI.
2. Tap **DO 0**, and three fields appear:
   - Name
   - Action Preset
   - Power Output
3. In the **Name** field, create the name and **Confirm**.
4. Tap the **Action Preset** field and nine choices are available.
5. Choose 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.

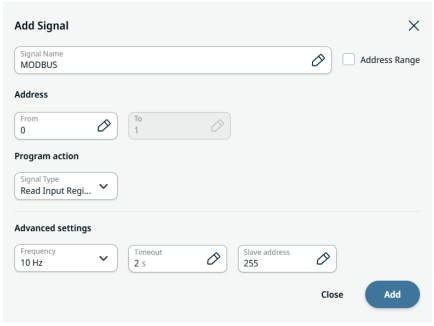
Modbus

Modbus is a serial communication protocol tied to a known IP address. It holds several input/output registers.

1. Tap **+ Add Source** under the Modbus tree, and the communication screen shows the Modbus: Source.



2. Tap the **IP address** field to edit and **Confirm**.
3. Check the **Sequential mode** checkbox to force the Modbus client to wait for a response before sending the next request.
4. Tap **+ Add Signal**, and a screen appears with these editable fields:
   - Signal Name
   - Address From
   - Address To
   - Signal Type
   - Frequency
   - Timeout
   - Slave Address



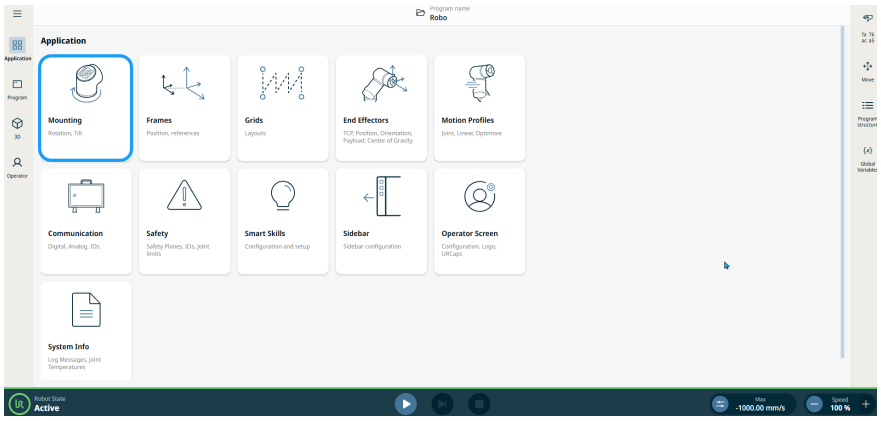
5. In the **Signal Name** field, you can edit and **Confirm**.
6. Check the **Address Range** checkbox to activate Address To field.
7. Enter the **Address From** and **To** respectively, and **Confirm**.
8. Tap the **Signal Type** field, and five choices are available:
    - Read Digital Inputs
    - Read Input Registers
    - Read Holding Registers
    - Write Digital Outputs
    - Write Output Registers
9. Choose the pertinent signal type.
10. Tap the **Frequency** field and choose the pertinent frequency.
11. Tap the **Timeout** field, enter the value, and **Confirm**.
12. On the **Slave Address** field, enter the value, and **Confirm**.
13. To finish, tap **Add** on the right bottom part of Add Signal screen.

15.2. 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.

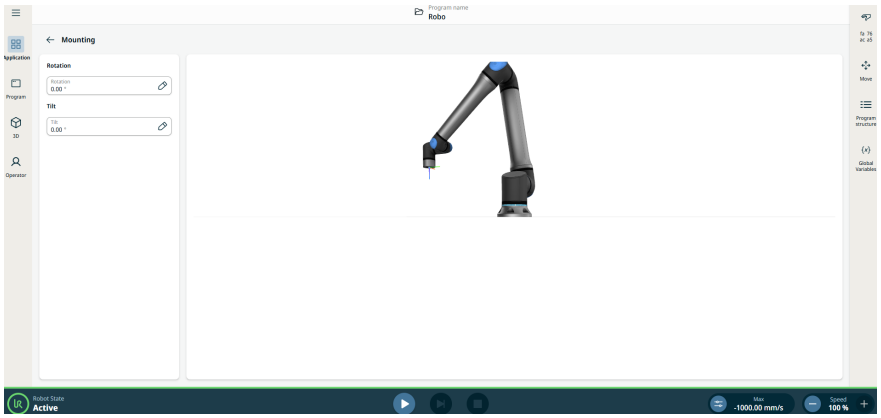


A screen appears with two fields on the left side:

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.

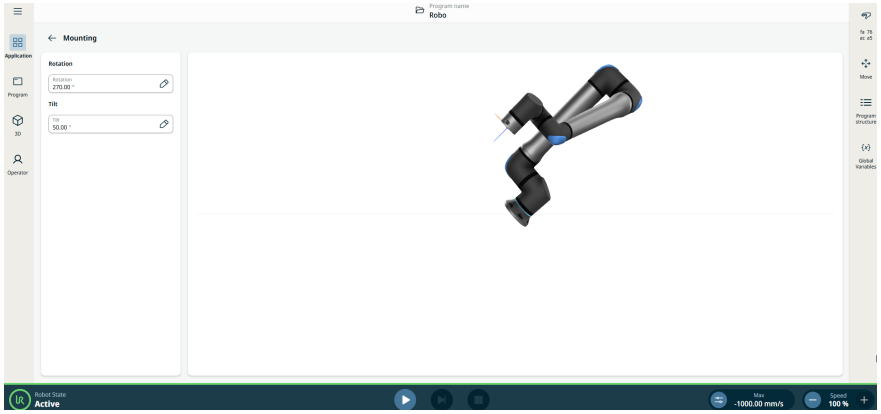


3. Use the following touch gestures for the robot arm.
   - **Press and rotate** clockwise or counterclockwise.

     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.

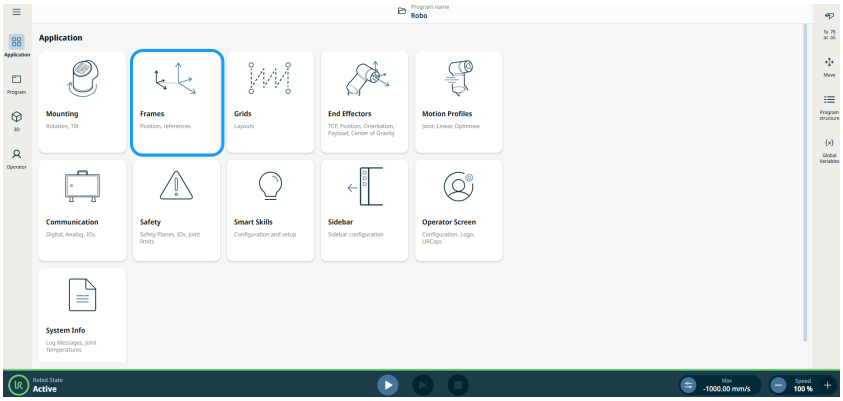


15.3. Frames

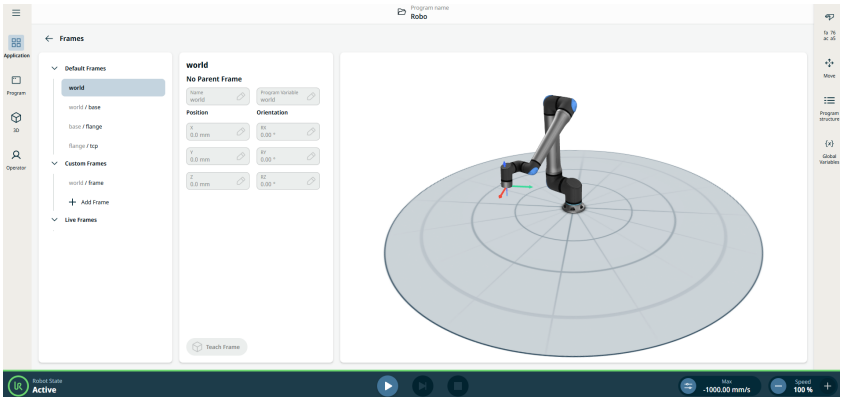
Description Frames application allows you to define frames of reference for use in a program.

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

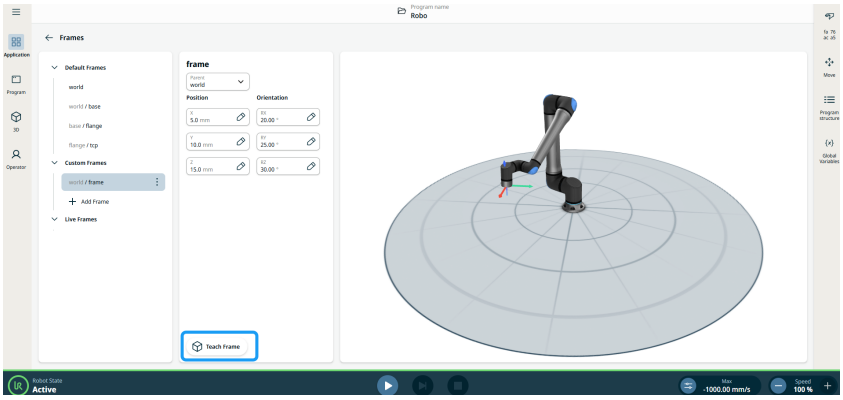
The right column/panel shows the robot arm in X, Y, Z coordinates.

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.

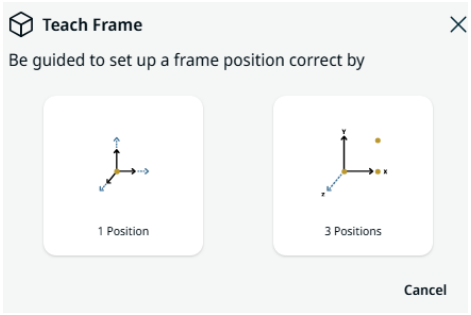
  Selecting any of the predefined frames will allow you to view their values. However, they cannot be edited.

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.

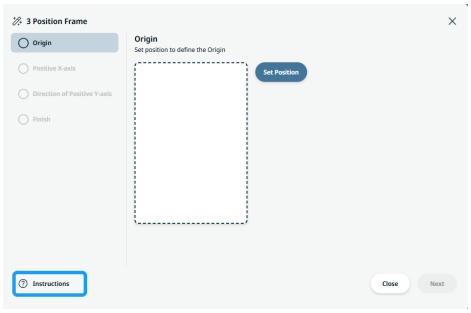


A pop-up screen appears with two options to set up a frame position.

- 1 Position
- 3 Positions

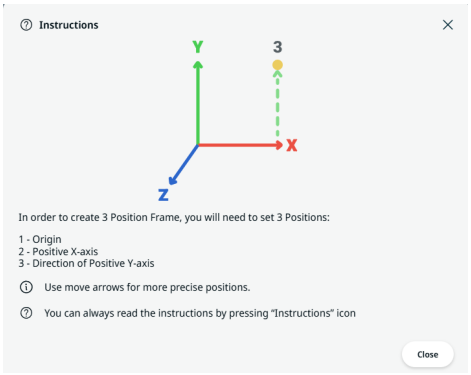


8. When you select **1 Position**, the 3D viewer becomes the main screen. Use the plus-minus button to translate and rotate the robot arm's three coordinates.
9. Tap **Save**.
10. When you select **3 Positions**, a pop-up screen appears. Tap the **Instructions** icon on the bottom left side.



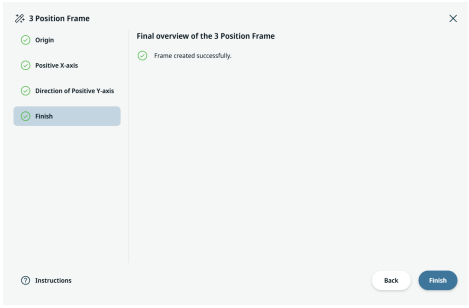
The Instructions box on how to create 3-Position Frame appears.

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.



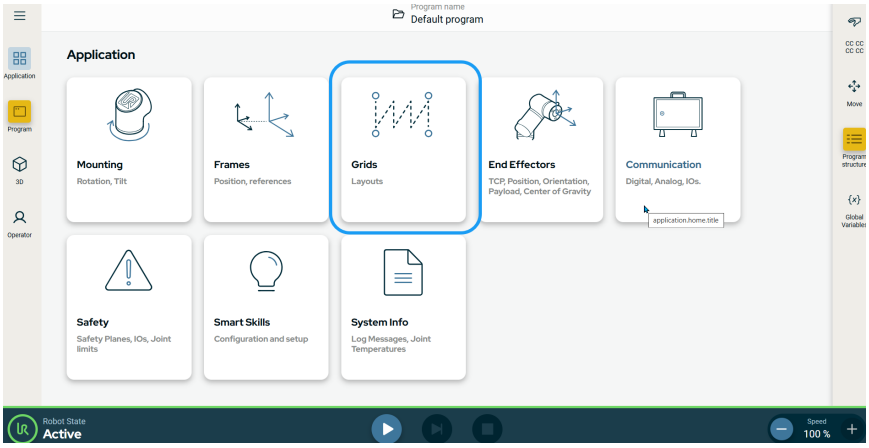
15.4. Grids Application

Description

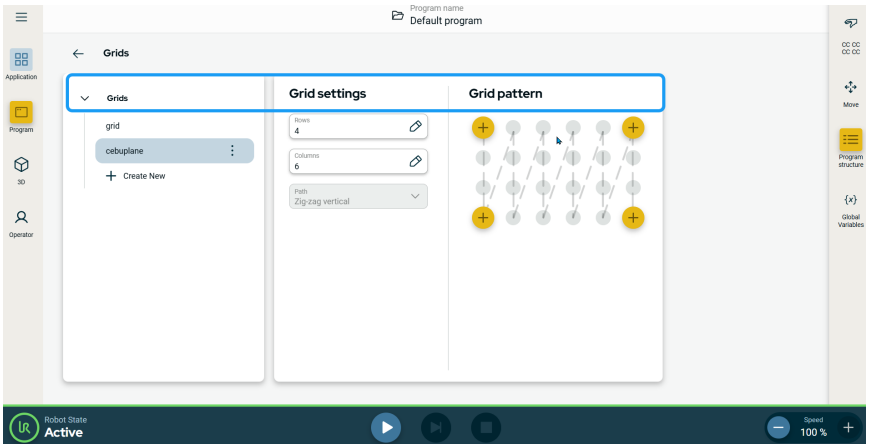
The **Grids** application is used to set up a grid, such as for palletizing, where you define four corners and the number of positions between them. This is like the palletizing wizard in PolyScope 5.

Using the Grids application functionality

1. Go to the Application menu screen. See Application Tab.
2. Tap the **Grids** icon.

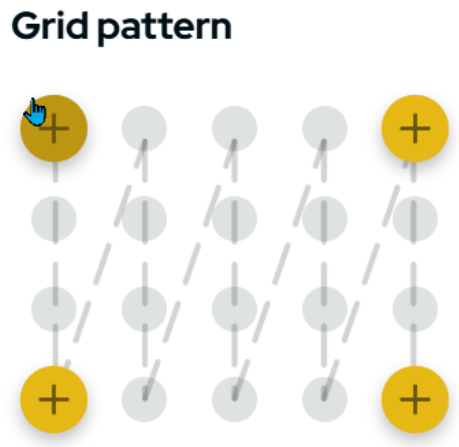


In the main screen, you can see the grids tree on the left panel, and the grid settings and grid pattern on the right panel.



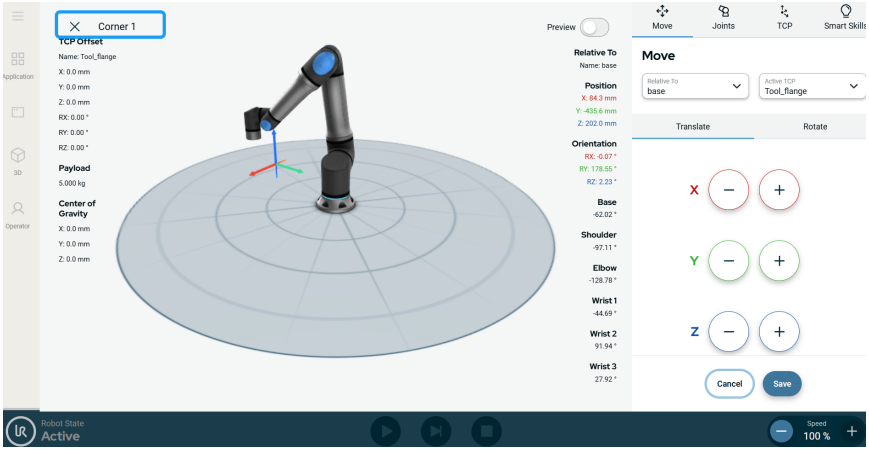
3. Tap **Create New** on the grids tree.
4. Enter the grid name in the **create new grid** field and **Save**.
5. Enter the number of **rows** and **columns** in the fields provided under the grid settings and tap **Confirm**. The grid pattern corresponds with what you input in the grid settings.

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.

15.5. 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:

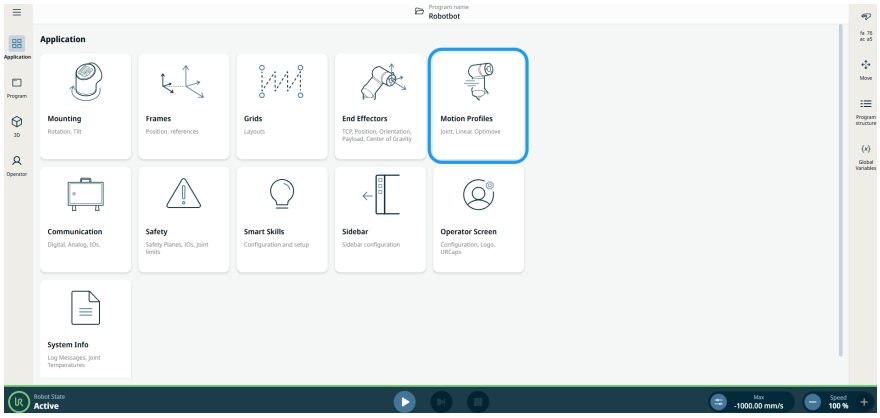
- Joint Move
- Linear/Direction Move
- Process/Circular Move

Intended programming flow

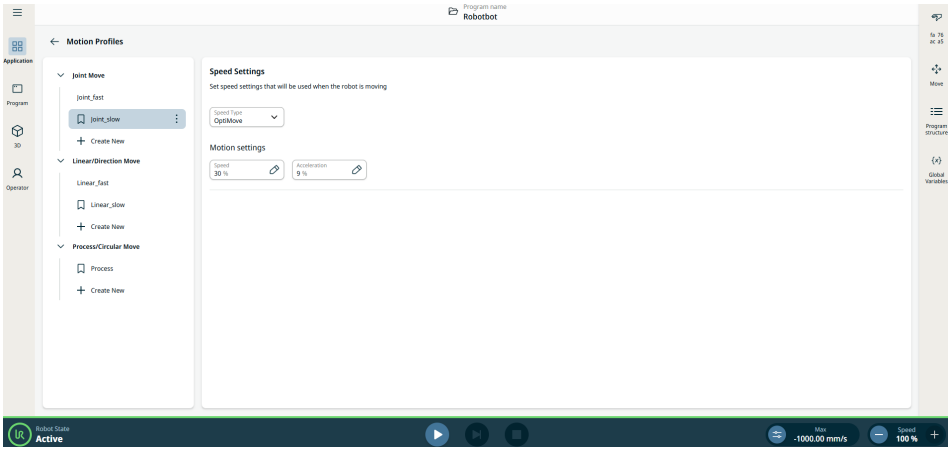
1. User starts by setting up all the Motion Profile needed for the application.
2. Choose a suitable name for easy recognition.
3. Choose the proper Move type (Joint, Linear/Direction, Process Move, Circular arc) from the commands toolbox when the Move node is inserted .
4. Every Move node should use a Motion Profile for easy overview.

Using the Motion Profiles application functionality

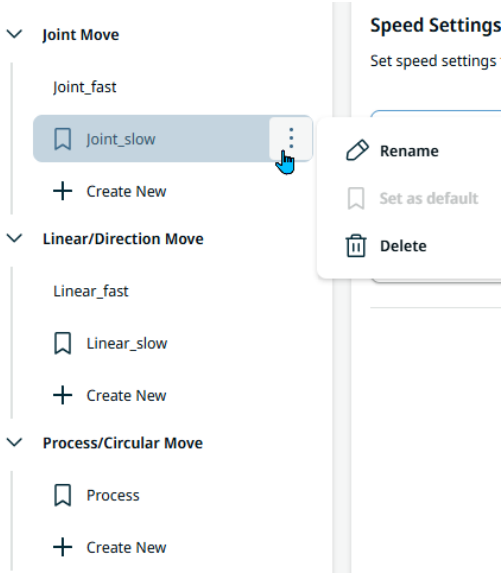
1. Go to the application nodes screen. See Application Tab.
2. Tap the **Motion Profiles** icon.



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:

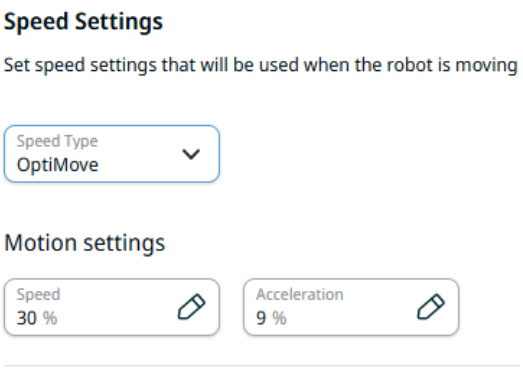
- Joint Move
- Linear/Direction Move
- Process/Circular Move

Note: Derived program nodes will use the profile for their underlying move type.
13.3.14 Joint Move on page 135 for joint profile.
13.3.6 Direction on page 119 node for linear profile.
13.3.4 Circular Arc Move on page 110 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.

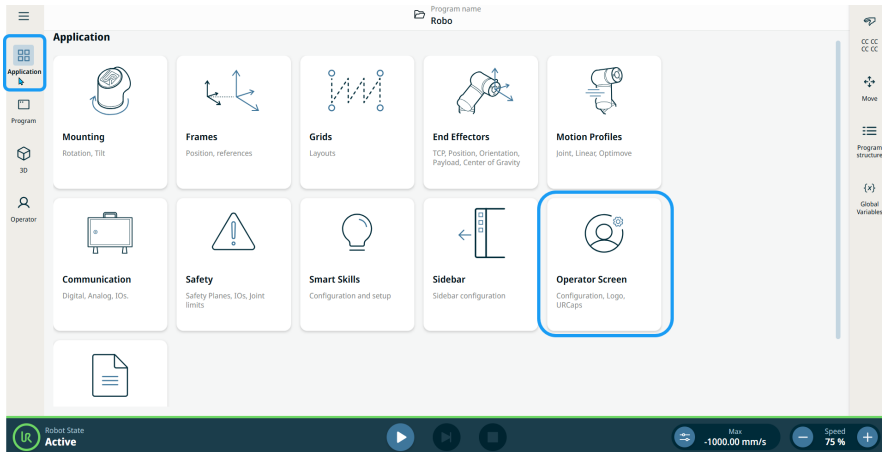
15.6. Operator Screen Application

Description

The **Operator Screen** application enables you to make changes in the contents of the Operator, located in the left toolbar. See Operator Screen.

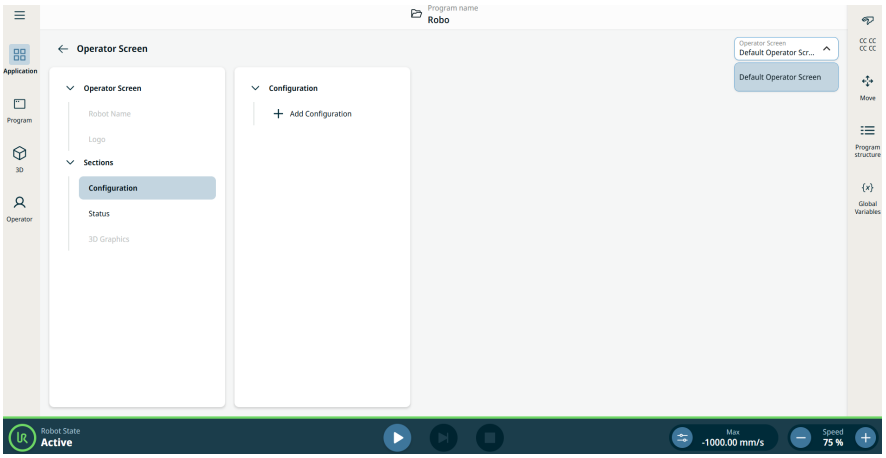
To access Operator Screen application

1. Tap the Application Tab in the left toolbar. The eleven application submenus appear in the main screen.
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.



16. 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.

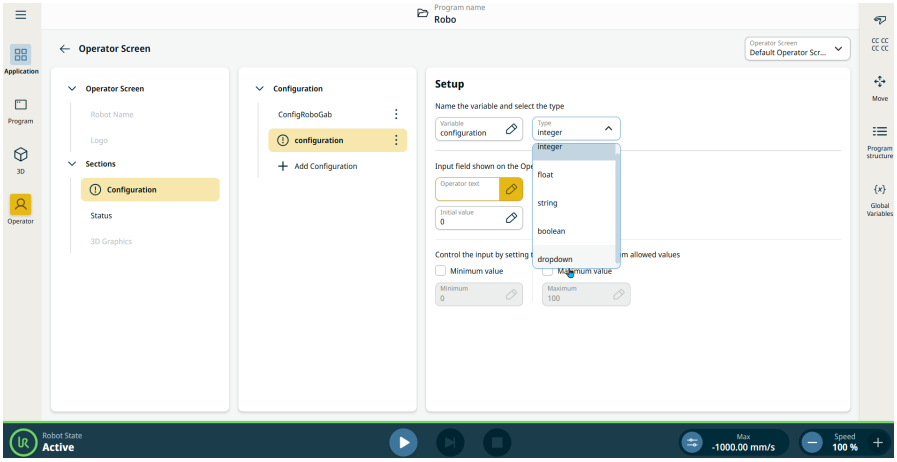
To configure Operator Screen

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.

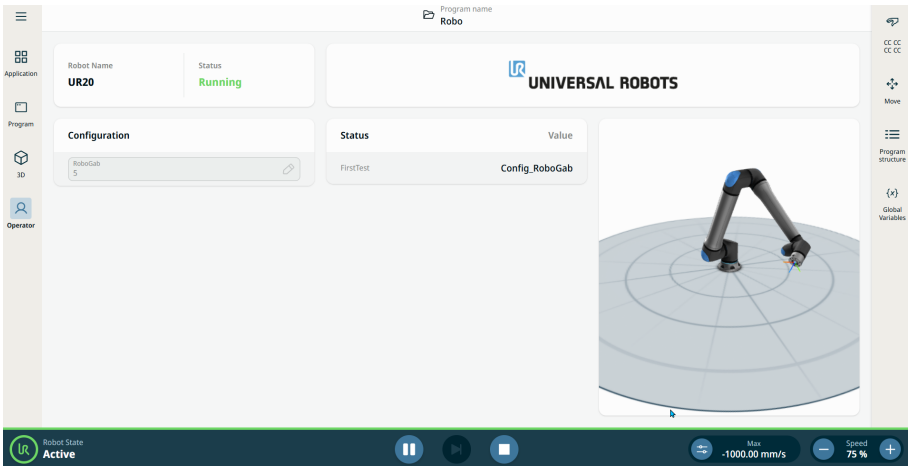


3. Tap the **Types** field, and four options are selectable:
   - Integer. For whole number.
   - Float. For decimal point value.
   - String. For text or a sequence of characters.
   - Boolean. Either true or false.
   - Dropdown

   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.



17. Services

17.1. Ethernet/IP

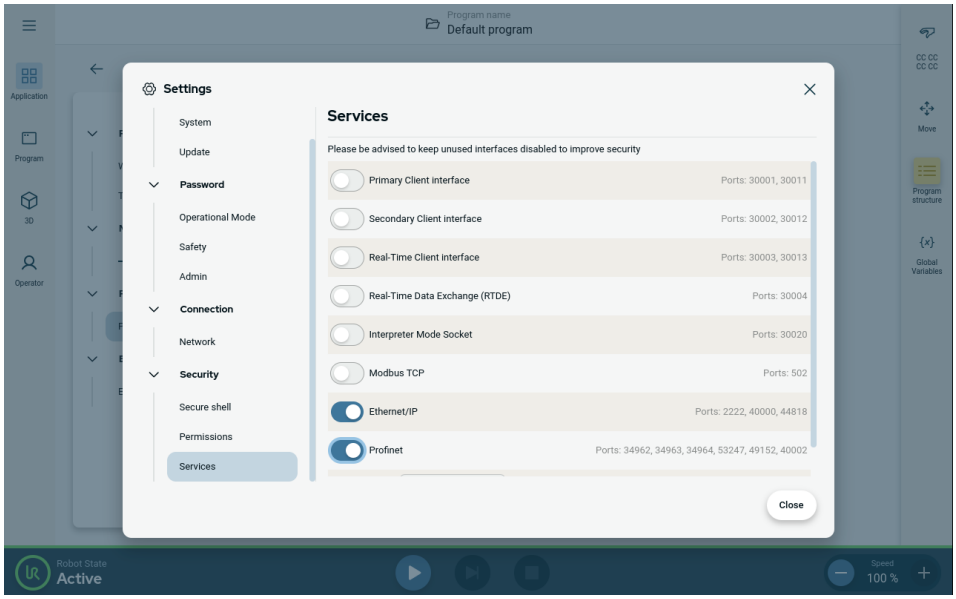
Description

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 you enable to Ethernet/IP function in PolyScope X.

1. In the top right of the screen, tap the Hamburger menu and then tap Settings.
2. In the menu on the left, under Security, tap Services.
3. Tap the Profinet button to switch Profinet on.



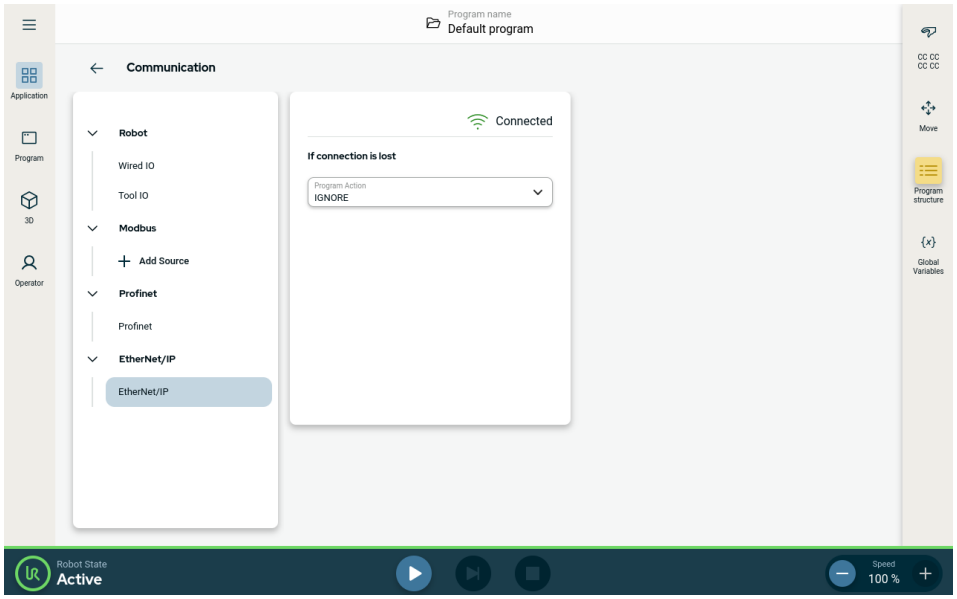
Using Ethernet/IP

Find the Ethernet/IP functions in PolyScope X:

In the PolyScope X left header.

1. Tap the Application icon.
2. Select the relevant action from the list.

Ignore	PolyScope X ignores the loss of EtherNet/IP connection, and the program continues to run.
Pause	PolyScope X pauses the current program. The program resumes from where it stopped.
Stop	PolyScope X stops the current 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 Device.
No Scanner	Ethernet/IP is running, but no device is connected to the robot via Ethernet/IP.
Disabled	Ethernet/IP is not enabled.

17.2. Profinet

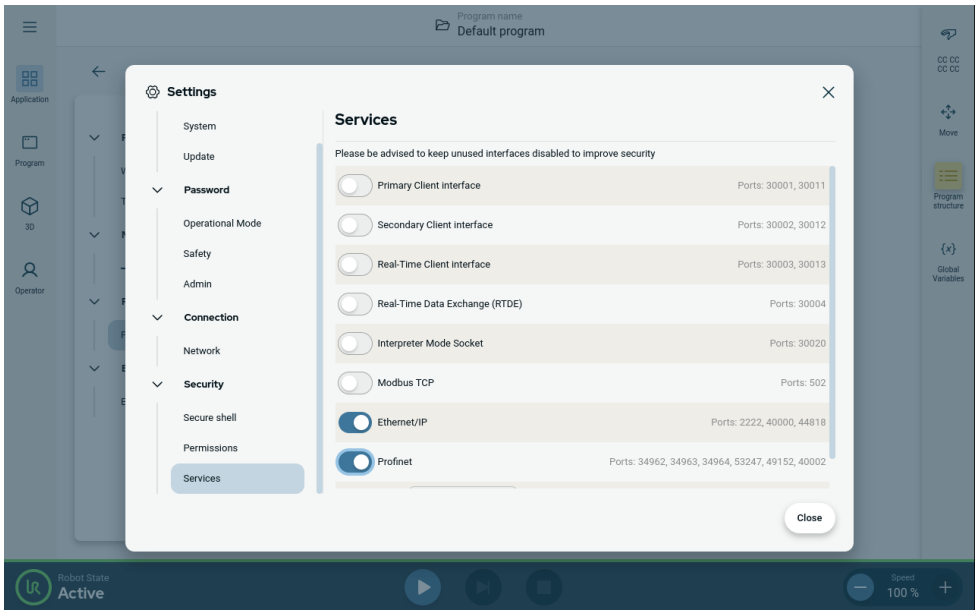
Description

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 to Profinet function in PolyScope X.

1. In the top right of the screen, tap the Hamburger menu and then tap Settings.
2. In the menu on the left, under Security, tap Services.
3. Tap the Profinet button to switch Profinet on.



Using Profinet

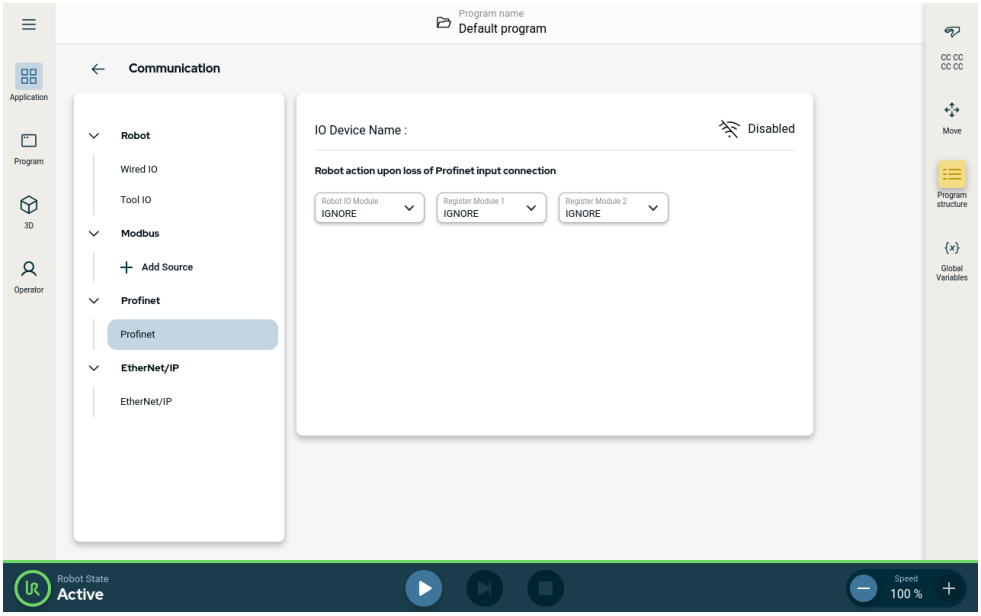
Find the profinet functions in PolyScope X:

In the PolyScope X left header.

1. Tap the Application icon.
2. Select Profinet from the left menu.

Select the relevant action from the list:

Ignore	PolyScope X ignores the loss of Profinet connection, and the program continues to run.
Pause	PolyScope X pauses the current program. The program resumes from where it stopped.
Stop	PolyScope X stops the current program.



Software Name: PolyScope X
Software Version: 10.10
Document Version: 10.13.349