10.5. The First Program

Description

A program is a list of commands telling the robot what to do. For most tasks, programming is done entirely using the PolyScope software. PolyScope allows you to teach the robot arm how to move using a series of waypoints to set up a path for the robot arm to follow.

Use the Move tab to move the Robot Arm to a desired position, or teach the position by pulling the Robot Arm into place while holding down the Freedrive button at the top of the Teach Pendant.

You can create a program can to send I/O signals to other machines at certain points in the robot's path, and perform commands like **if...then** and **loop**, based on variables and I/O signals.

To create a simple program

1. On PolyScope, in the Header **File Path**, tap **New...** and select **Program**.
2. Under Basic, tap **Waypoint** to add a waypoint to the program tree. A default MoveJ is also added to the program tree.
3. Select the new waypoint and in the Command tab, tap **Waypoint**.
4. On the Move Tool screen, move the robot arm by pressing the move arrows. You can also move the robot arm by holding down the Freedrive button and pulling the Robot Arm into desired positions.
5. Once the robot arm is in position, press **OK** and the new waypoint displays as Waypoint_1.
6. Follow steps 2 to 5 to create Waypoint_2.
7. Select Waypoint_2 and press the Move Up arrow until it is above Waypoint_1 to change the order of the movements.
8. Stand clear, hold on to the emergency stop button and in the PolyScope Footer, press **Play** button for the Robot Arm to move between Waypoint_1 and Waypoint_2. Congratulations! You have now produced your first robot program that moves the Robot Arm between the two given waypoints.

NOTICE

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

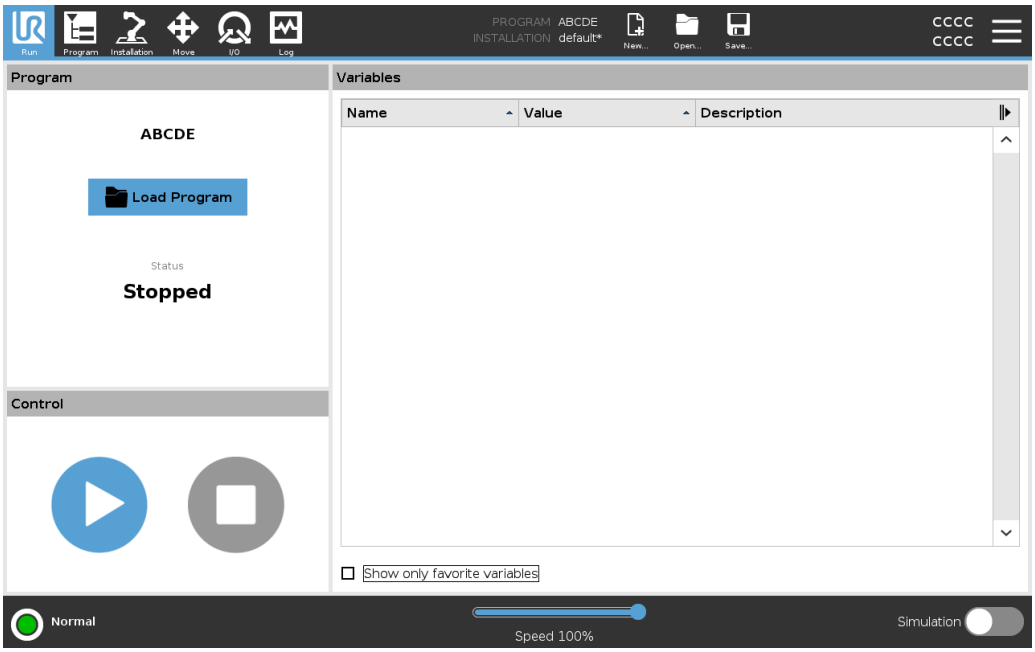
WARNING

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

10.5.1. Run Tab

Description

The **Run** tab allows you to do simple operations and monitor the state of your robot. You can load, play, pause and stop a program, as well as monitor variables. The Run Tab is most useful when the program is created and the robot is ready for operation.



Program

The Program pane displays the name and status of the current program.

To load a new program

1. In the Program pane, tap **Load Program**.
2. Select your desired program from the list.
3. Tap **Open** to load the new program.

   The variables, if present, are displayed when you play the program.

Variables

The Variables pane displays the list of variables used by programs to store and update values during runtime.

- Program variables belong to programs.
- Installation variables belong to installations that can be shared among different programs. The same installation can be used with multiple programs.

All program variables and installation variables in your program are displayed in the Variables pane as a list showing the Name, Value and Description of the variable.

Variable descriptions

You can add information to your variables by adding variable descriptions in the Description column. You can use the variable descriptions to convey the purpose of the variable and/or the meaning of its value to operators using the Run tab screen and/or other programmers.
Variable descriptions (if used) can be up to 120 characters, displayed in the Description column of the variables list on the Run tab screen and the Variables tab screen.

Favorite variables

You can display selected variables by using the **Show only favorite variables** option.
To show favorite variables

1. Under Variables, check the **Show only favorite variables** box.
2. Check **Show only favorite variables** again to show all variables.

You cannot designate favorite variables in the Run Tab, you can only display them. Designating favorite variables depends on the variable type.

To designate favorite program variables

1. In the Header, tap **Program**.
   The variables are listed under **Variable Setup**.
2. Select the desired variables.
3. Check the **Favorite variable** box.
4. Tap **Run** to return to your variable display.

To designate favorite installation variables

1. In the Header, tap **Installation**.
2. Under General, select **Variables**.
   The variables are listed under **Installation Variables**.
3. Select the desired variables.
4. Check the **Favorite variable** box.
5. Tap **Run** to return to your variable display.

Collapse/expand the Description column

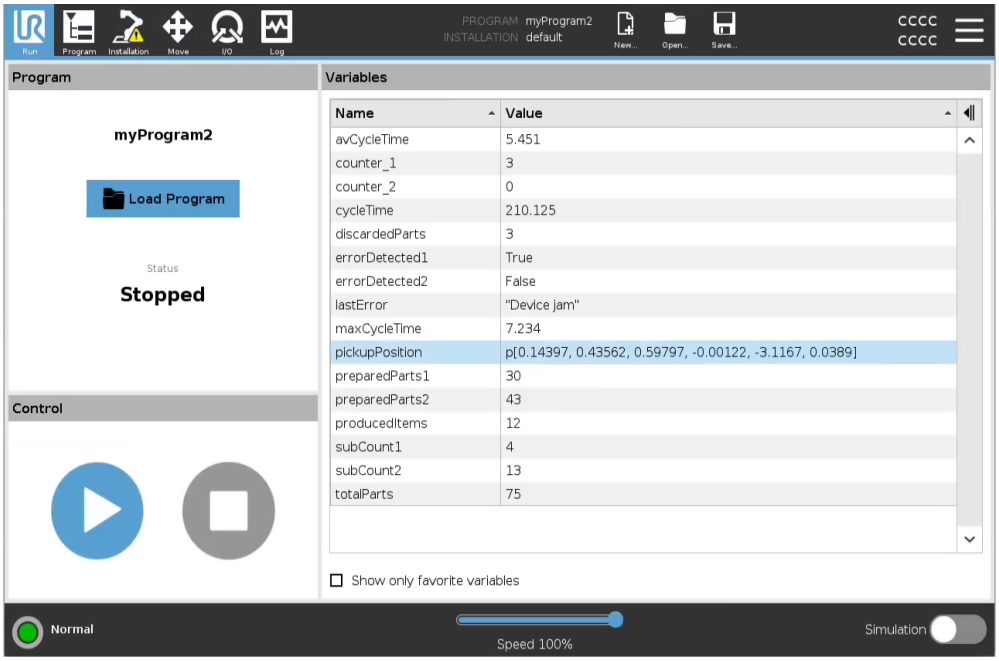
A variable description spans multiple lines to fit the width of the Description column if necessary. You can also collapse and expand the Description column by using the buttons shown below.

To collapse/expand the Description column

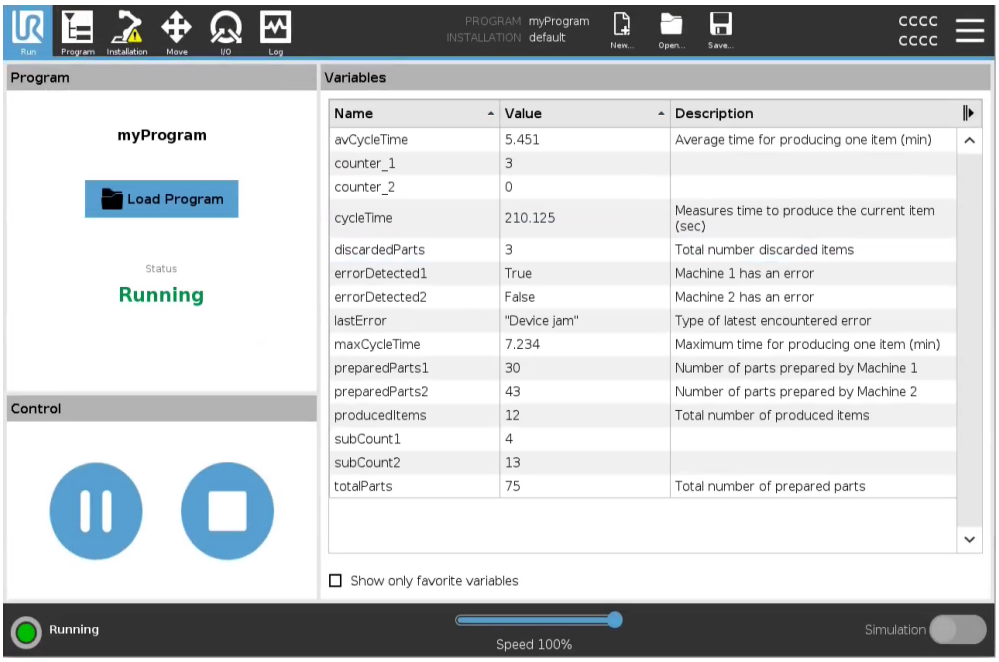
1. Tap  to collapse the Description column.
2. Tap  to expand the Description column.

here

Collapsed Description column



Expanded Description column



Control The Control pane allows you to control the running program. You can play and stop, or pause and resume a program, using the buttons listed in the table below:

- The Play button, Pause button and the Resume Button are combined.
- The Play button changes to Pause when the program is running.
- The Pause button changes to Resume.

Button		Function
Play		To play a program 1. Under Control, tap **Play** to start running a program from the beginning.
Resume		To resume a paused program 1. Tap **Resume** to continue running the paused program.
Stop		To stop a program 1. Tap **Stop** to stop the running program You cannot resume a stopped program. You can tap **Play** to restart the program.
Pause		To pause a program 1. Tap **Pause** to pause a program at a specific point. You can resume a paused program.

10.5.2. Move Robot into Position

Description

Access the **Move Robot into Position** screen when the Robot Arm must move to a particular start position before running a program, or when the Robot Arm is moving to a waypoint while modifying a program.

In cases where the **Move Robot into Position** screen cannnot move the Robot Arm to the program start position, it moves to the first waypoint in the program tree.
The Robot Arm can move to an incorrect pose if:

- The TCP, feature pose or waypoint pose of the first movement is altered during program execution before the first move is executed.
- The first waypoint is inside an If or Switch program tree node.

Accessing the Move Robot into Position Screen

1. Tap the Run tab in the header.
2. In the **Footer**, tap **Play** to access the **Move Robot into Position** screen.
3. Follow the on-screen instructions to interact with the animation and the real robot.

Move robot to

Hold down **Move robot to:** to move the Robot Arm to a start position. The animated Robot Arm displayed on-screen shows the desired movement about to be performed.

NOTICE

Collision can damage the robot or other equipment. Compare the animation with the position of the real Robot Arm to ensure the Robot Arm can safely perform the movement without colliding with any obstacles.

Manual

Tap **Manual** to access the **Move** screen where the Robot Arm can be moved by using the Move Tool arrows and/or configuring Tool Position and Joint Position coordinates.

10.5.3. Using the Program Tab

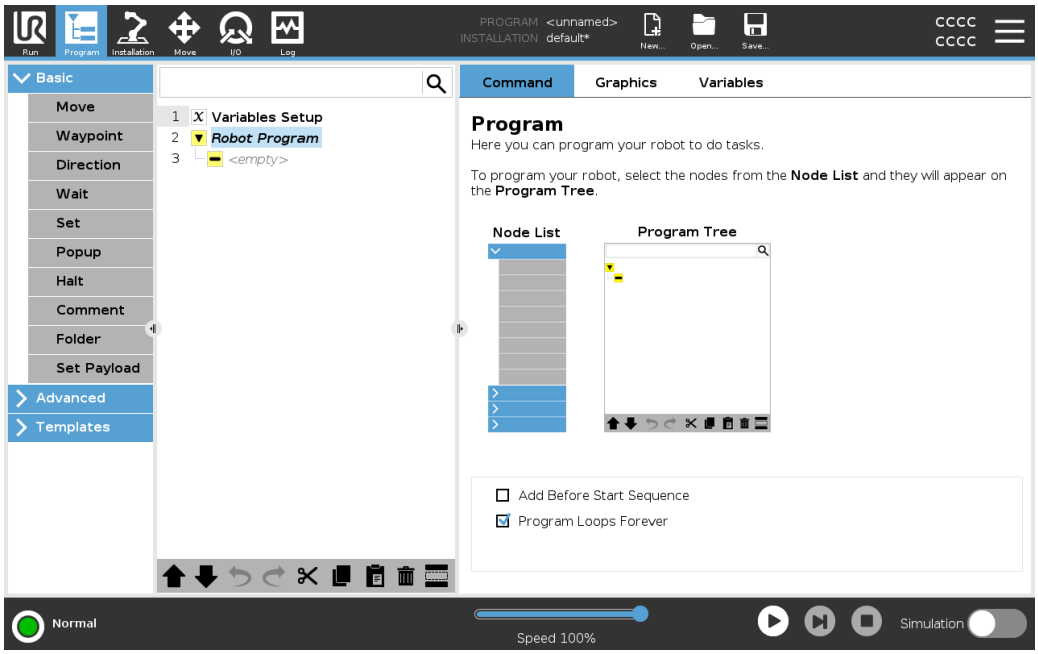
Description

The Program tab is the where you create and edit robot programs. There are two main areas:

- The left side contains the program nodes you can add to your robot program.

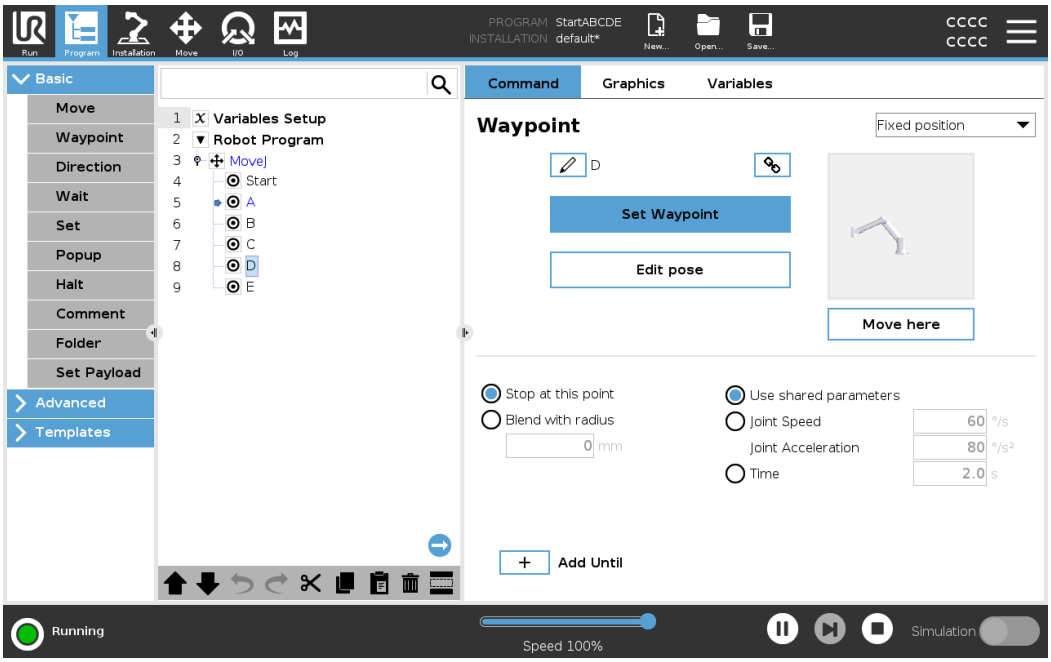
  You can use the Basic, Advanced and Template dropdowns to the very left.
- The right side contains the configuration of the program nodes you can add to your program.

  You can use Command, Graphics and Variables options.



Program Tree

The program tree is built as you add program nodes to your program.
You can use the Command tab to configure the functionality of the added program nodes.

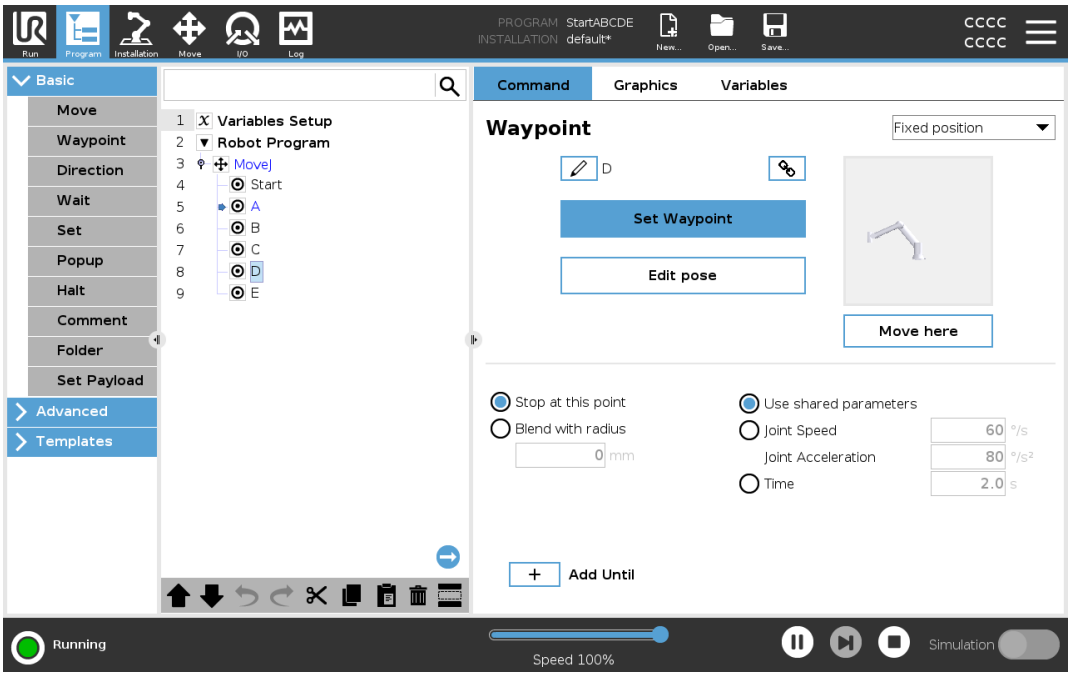


Adding program nodes

- You cannot run an empty program tree or a program containing incorrectly configured program nodes.
- Incorrectly configured programs nodes are higlighted in yellow.
- Correctly configured program nodes are highlighted in white.

Program Execution Indication

Robot programs often become quite long, so order to be able to see the flow of the robot program, you can look at what program node is active.



When the program is running, the program node currently being executed is indicated by a small icon next to that node.
The path of execution is highlighted with blue arrow .

Tapping the icon at the corner of the program allows it to track the command being executed

Search Button

You can also search for a specific command/program node. This is useful when you have a long program with many different program nodes.

10.5.4. Program Tree Toolbar

Description

You can work with the program nodes that have been added to the program tree by using the icons in the bottom of the program tree.

Icons in the Program Tree toolbar

Use the toolbar at the base of the Program Tree to modify the Program Tree.

Undo & Redo		undo and redo changes to commands.
Move Up & Move Down		changes the position of a node.
Cut		cuts a node and allows it to be used for other actions (e.g., paste it on other place on the Program Tree).
Copy		copies a node and allows it to be used for other actions (e.g., paste it on other place on the Program Tree).
Paste		pastes a node that was previously cut or copied.
Delete		removes a node from the Program Tree.
Suppress		suppresses specific nodes on the Program Tree.
Search Button		search in the Program Tree. Tap the  icon to exit search.

10.5.5. Using Selected Program Nodes

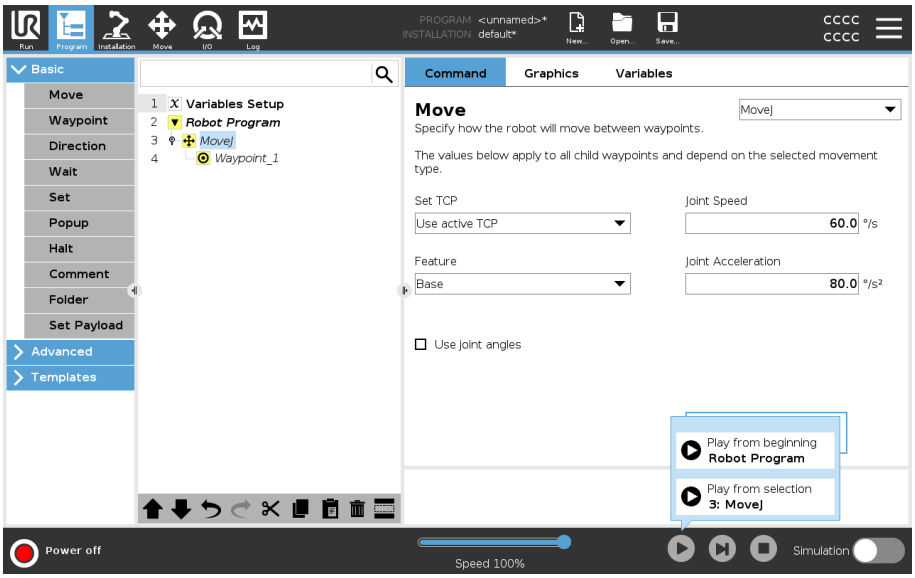
Description

You can start your robot program from any program node in the program tree. This is useful when you are testing your program.

When the robot is in Manual Mode (see Operational Modes), you can allow a program to start from a selected node or you can start the entire program from the beginning.

Play From Selection

The Play button in the Footer provides options for how to start the program.
In the image below, the **Play** button is selected and **Play from Selection** is displayed.



- You can start a program only from a node in the robot Program tree. The **Play from Selection** stops if a program cannot be run from a selected node.

  The program also stops and displays an error message if an unassigned variable in encountered while playing a program from selected node.
- You can use **Play from Selection** in a subprogram. The program execution halts when the subprogram ends.
- You cannot use **Play from Selection** with a thread because threads always start from the beginning.

To play a program from a selected node

1. In the Program tree, select a node.
2. In the Footer, tap **Play**.
3. Select **Play from Selection** to run a program from a node in the program tree.

Example

You can start a stopped program again from a specific node.

10.5.6. Using Basic Program Nodes

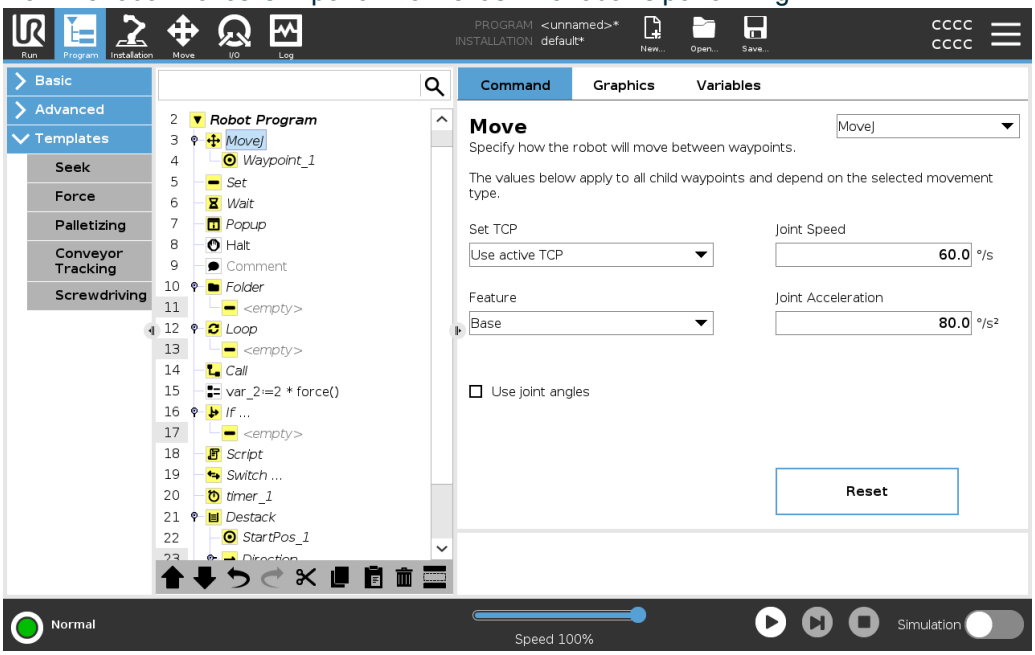
Description

Basic program nodes are used to create simple robot applications. Some basic program nodes are also used to organize your robot program and create comments in your robot program. This can be quite useful, if it is large robot program.

10.5.7. Basic Program Nodes: Move

Description

The Move command allows the robot to move from point A to point B.
How the robot moves is important to the task the robot is performing.



When you add a Move to your program tree, the Move pane appears to the right of the screen.

The Movecommand controls the robot's motion via waypoints.
Waypoints are automatically added when you add Move commands to a program.
Read more about Waypoints.
You can also use Moves to set acceleration and speed for the robot arm's movement between waypoints.

The robot moves using four Move commands. See the Move command types below:

MoveJ

The MoveJ 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.

Add a MoveJ command

1. In your robot program, select the place where you wish to add a Move.
2. Under Basic, tap **Move** to add a waypoint to the robot program together with a Move node.
3. Select the move node.
4. Select the MoveJ in the drop-down menu.

Detail

MoveJ 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. The shared parameters that apply to this movement type are the maximum joint speed and joint acceleration, specified in *deg/s* and deg/s^2, respectively. If it is desired to have the robot arm move fast between waypoints, disregarding the path of the tool between those waypoints, this movement type is the preferable choice.

MoveL

The MoveL command creates a movement that is a direct line from point A and point B.

Add a MoveL command

1. In your Robot Program, select the place where you wish to add a Move.
2. Under Basic, tap **Move** to add a waypoint to the robot program together with a Move node.
3. Select the move node.
4. Select the MoveL from the drop-down menu.

Detail

MoveL 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. The shared parameters that can be set for this movement type are the desired tool speed and tool acceleration specified in *mm/s* and mm/s^2, respectively, and also a feature.

MoveP

The MoveP command creates a movement with a constant speed between the waypoints. Blend between waypoints is enabled to ensure constant speed. (See Blending).

Add a MoveP command

1. In your Robot Program, select the place where you wish to add a Move.
2. Under Basic, tap **Move** to add a waypoint to the robot program together with the Move node.
3. Select the move node.
4. Select the MoveP from the drop-down menu.

Detail

MoveP moves the tool linearly with constant speed with circular blends, and is intended for some process operations, like gluing or dispensing. The size of the blend radius is by default a shared value between all the waypoints. A smaller value will make the path turn sharper whereas a higher value will make the path smoother. While the robot arm is moving through the waypoints with constant speed, the robot control box cannot wait for either an I/O operation or an operator action. Doing so might stop the robot arm's motion, or cause a robot stop.

MoveCircle

The MoveCircle command creates a circular movement, by creating a half circle. You can only add CircleMove via a MoveP command.

Add a MoveCircle command

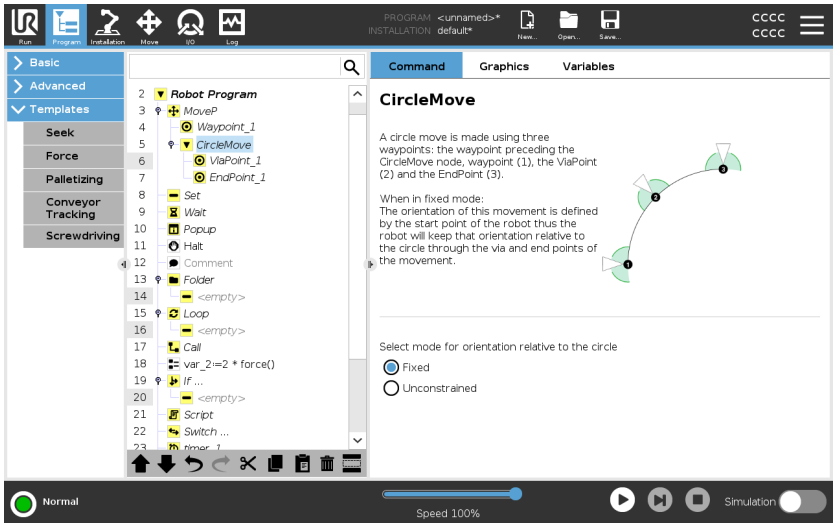
1. In your Robot Program, select the place where you wish to add a Move.
2. Under Basic, tap **Move**.

   A waypoint is added to the robot program together with the Move node.
3. Select the move node.
4. Select the MoveP from the drop-down menu.
5. Tap **Add circle move**
6. Select the orientation mode.

Detail

The robot starts the circular movement from its current position, or start point, and moves through a ViaPoint specified on the circular arc, to an EndPoint that completes the circular movement.
A mode is used to calculate tool orientation, through the circular arc.
The mode can be:

- Fixed: only the start point is used to define the tool orientation.
- Unconstrained: the start point transforms to the EndPoint to define tool orientation.



10.5.8. Basic Program Nodes: Waypoints

Description

Waypoints are one of the most central parts of a robot program, telling the robot arm where to go one movement at a time.

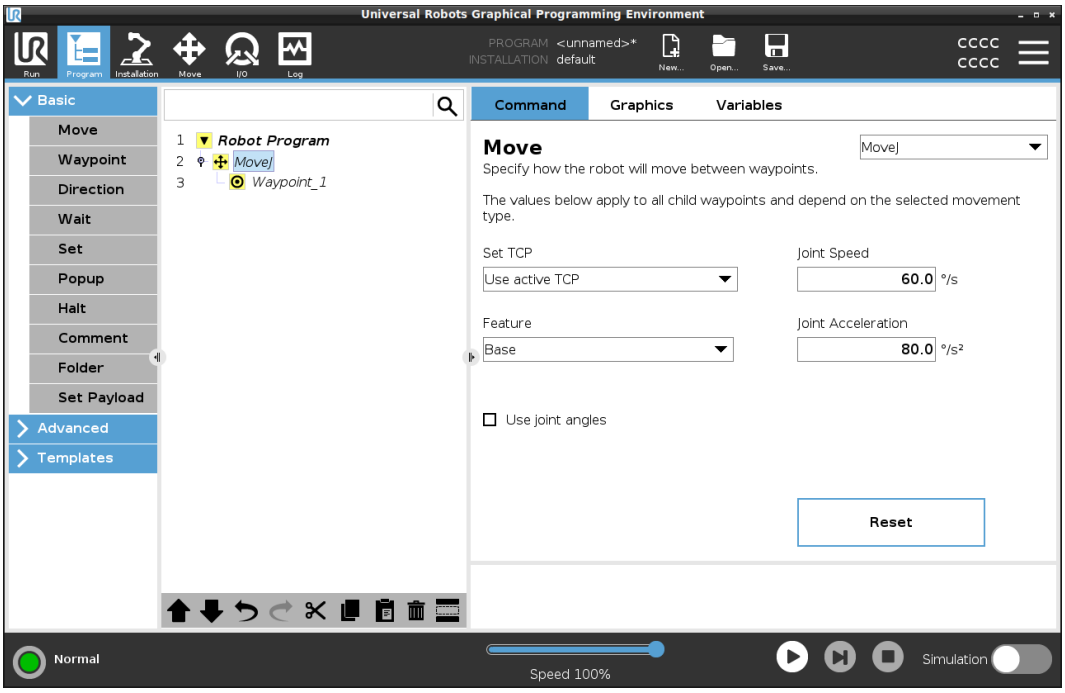
Add Waypoints

A waypoint accompanies a Move, so adding a Move is required for the first waypoint.

Add a waypoint to a robot program

1. In your Robot Program, select the place where you wish to add a Move.
2. Under Basic, tap **Move**.

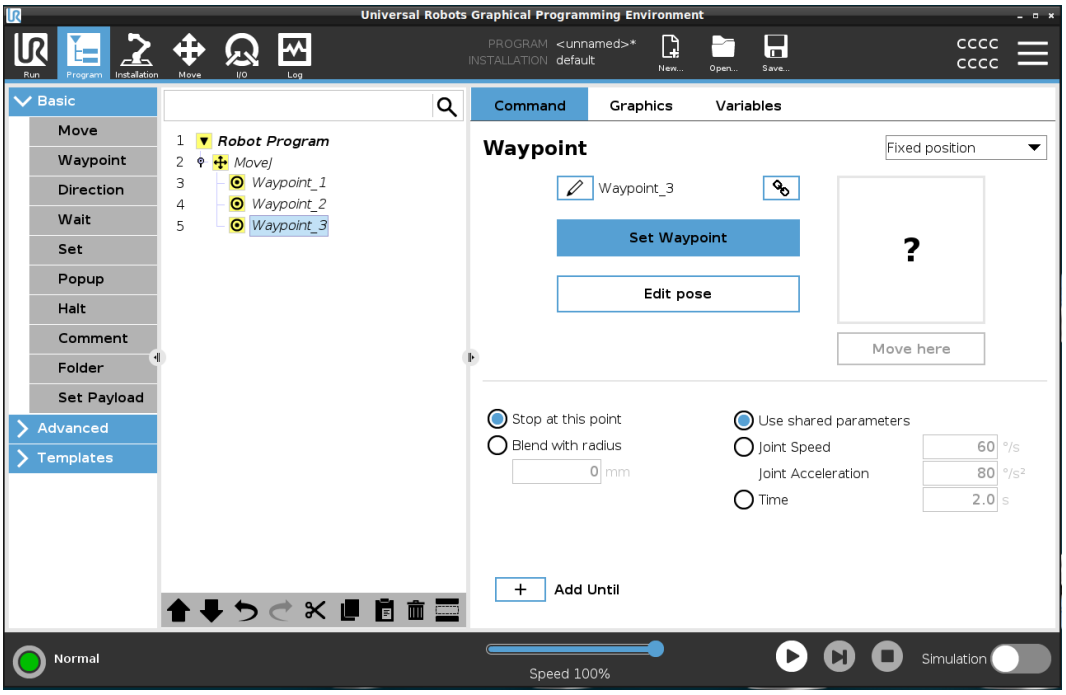
   A waypoint is added to the robot program together with the Move node.



Add additional waypoints to a Move or Waypoint

1. In your Robot Program, select a Move node or Waypoint node.
2. Under Basic, tap **Waypoint**.

   The additional waypoint is added in the Move node. This waypoint is part of the Move command.



The additional waypoint is added under the waypoint that you selected in the robot program.

Detail

Using a waypoint means applying the taught relationship between the feature and the TCP from the Move command. The relationship between the feature and the TCP, applied to the current selected feature, achieves the desired TCP location. The robot calculates how to position the arm to allow the current active TCP to reach the desired TCP position.

10.5.9. Set Payload

Description

The Set Payload command allows you to configure the payload for the robot. Payload is the combined weight of everything attached to the robot tool flange.
When to use:

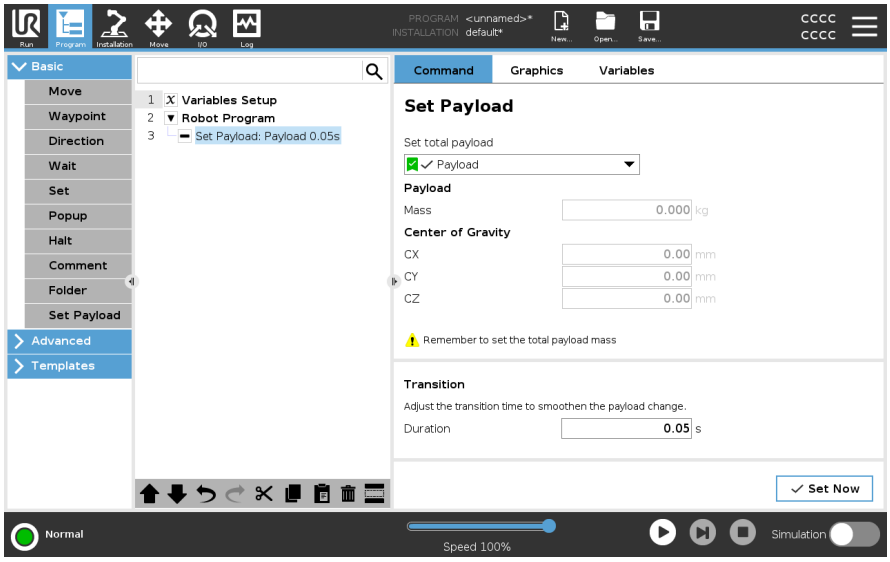
- When adjusting the payload weight to prevent the robot from triggering a robot stop. A correctly configured payload weight ensures optimal robot movement.

  Setting the payload correctly ensures optimal motion performance and avoids robot stops.
- When setting up the payload for use in a pick and place program, using a gripper.

Set Payload

Use the Set Payload command

1. In your robot program, select the place or node where you wish to add a Set command.
2. Under Basic, tap **Set Payload**.
3. Use the drop-down, under **Select Payload**.
   a. Select one of the payloads already configured.
   b. Or, use the drop-down to configure a new payload by selecting **Custom Payload** and completing the mass and CoG fields.



Tip You can also use the **Set Now** button to set the values on the node as the active payload.

Use tip Remember to always update your payload when making any changes to the configuration of the robot program.

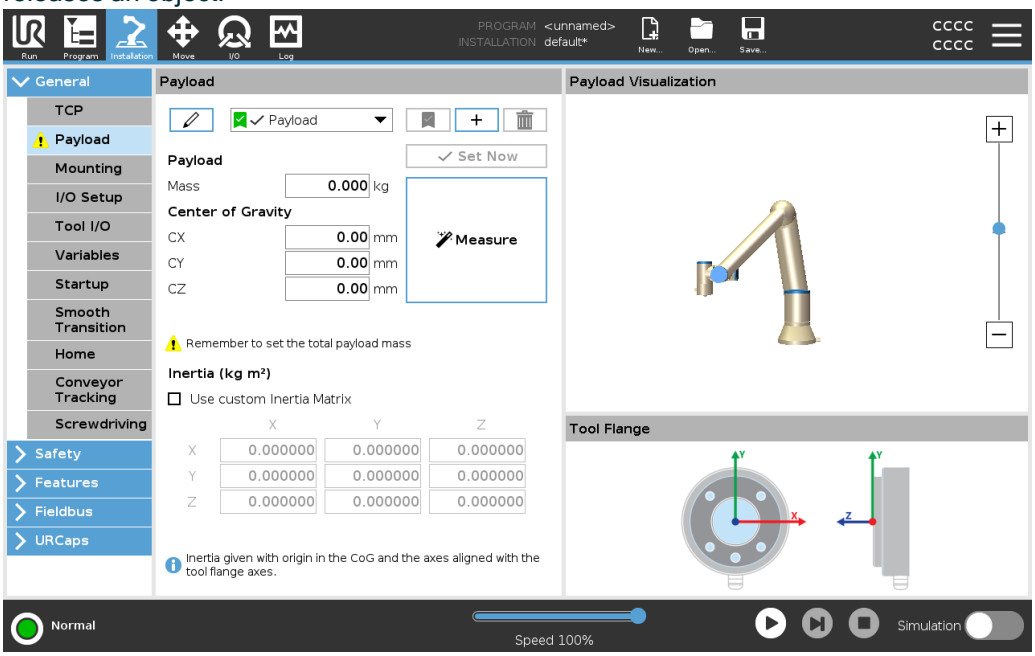
Example: Set Payload In a pick and place program, you would create a default payload in the installation. Then you add a Set Payload when picking up an object. You would update the payload after the gripper closes, but before starting to move.
Additionally, you would use the Set Payload after the object has been released.

Payload Transition Time This is the time it takes the robot to adjust for a given payload. At the bottom of the screen, you can set the transition time between different payloads.
You can add a payload transition time in seconds.
Setting a transition time larger than zero, prevents the robot from doing a small "jump", when the payload changes. The program continues while the adjustment is taking place.
Using the Payload Transition Time is recommended when picking up or releasing heavy objects or using a vacuum gripper.

10.5.10. Payload

Description

You must set the Payload, the CoG and the inertia for the robot to perform optimally. You can define multiple Payloads, and switch between them in your program. This is useful in Pick and Place applications, for example, where the robot picks up and releases an object.



Adding, Renaming, Modfying and Removing Payloads

You can start configuring a new Payload with the following actions:

- Tap the + to define a new Payload with a unique name. The new payload is available in the drop-down menu.
- Tap the to rename a Payload.
- Tap the to remove a selected Payload. You cannot remove the last Payload.

Active Payload

The checkmark in the drop-down indicates which payload is active ✓ Payload. The active Payload can be changed using the ✓ Set Now.

Default Payload

The default Payload is set as the active Payload before the program starts.

- Select the desired Payload and tap **Set as default** to set a Payload as the default.

The green icon in the drop-down menu indicates the default configured Payload ✓ Payload.

Setting the Center of Gravity

Tap the fields CX, CY and CZ to set the center of gravity. The settings apply to the selected Payload.

Payload Estimation

This feature allows the robot to help set the correct Payload and Center of Gravity (CoG).

Using the Payload Estimation Wizard

1. In the Installation Tab, under General, select **Payload**.
2. On the Payload screen, tap **Measure**.
3. In the Payload Estimation Wizard tap **Next**.
4. Follow the steps in the Payload Estimation Wizard to set the four positions. Setting the four positions requires moving the robot arm into four different positions. The load of the payload is measured at each position.
5. Once all measurements are complete, you can verify the result and tap **Finish**.

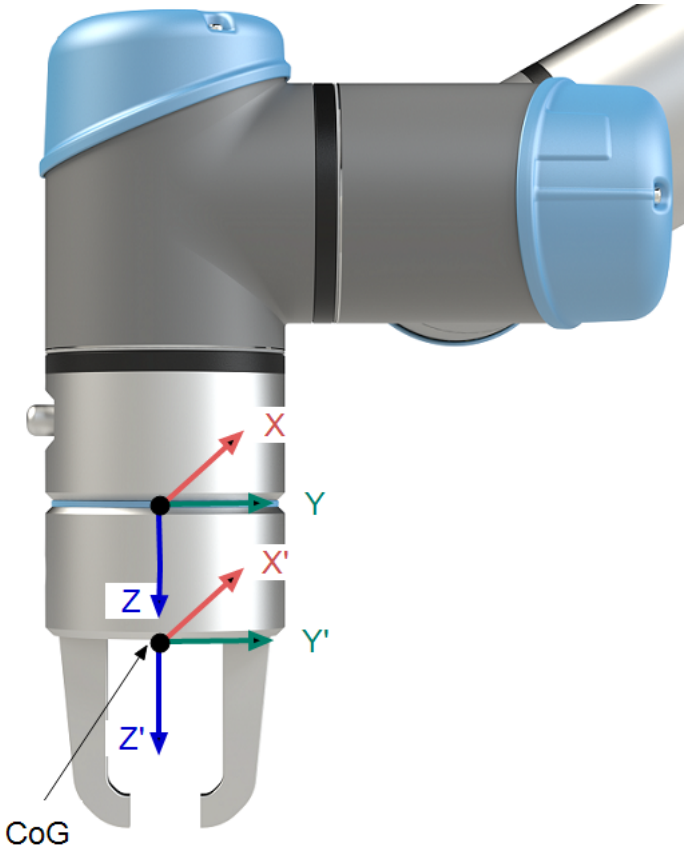
NOTICE

Follow the these guidelines for best Payload Estimation results:

- Ensure the TCP positions are as different as possible from each other
- Perform the measurements within a short timespan
- Avoid pulling on the tool and/or attached payload before and during estimation
- Robot mounting and angle must be correctly defined in the installation

Setting Inertia Values

You can select **Use custom Inertia Matrix** to set inertia values.
Tap the fields: `IXX`, `IYY`, `IZZ`, `IXY`, `IXZ` and `IYZ` to set the inertia for the selected Payload.
The inertia is specified in a coordinate system with the origin at the Center of Gravity (CoG) of the payload and the axes aligned with the tool flange axes.
The default inertia is calculated as the inertia of a sphere with the user specified mass, and a mass density of $1g/cm^3$



10.5.11. Mounting

Description

Specifying the mounting of the Robot arm serves two purposes:

1. Making the Robot arm appear correctly on screen.
2. Telling the controller about the direction of gravity.

An advanced dynamics model gives the Robot arm smooth and precise motions, as well as allows the Robot arm to hold itself in **Freedrive Mode**. For this reason, it is important to mount the Robot arm correctly.

WARNING

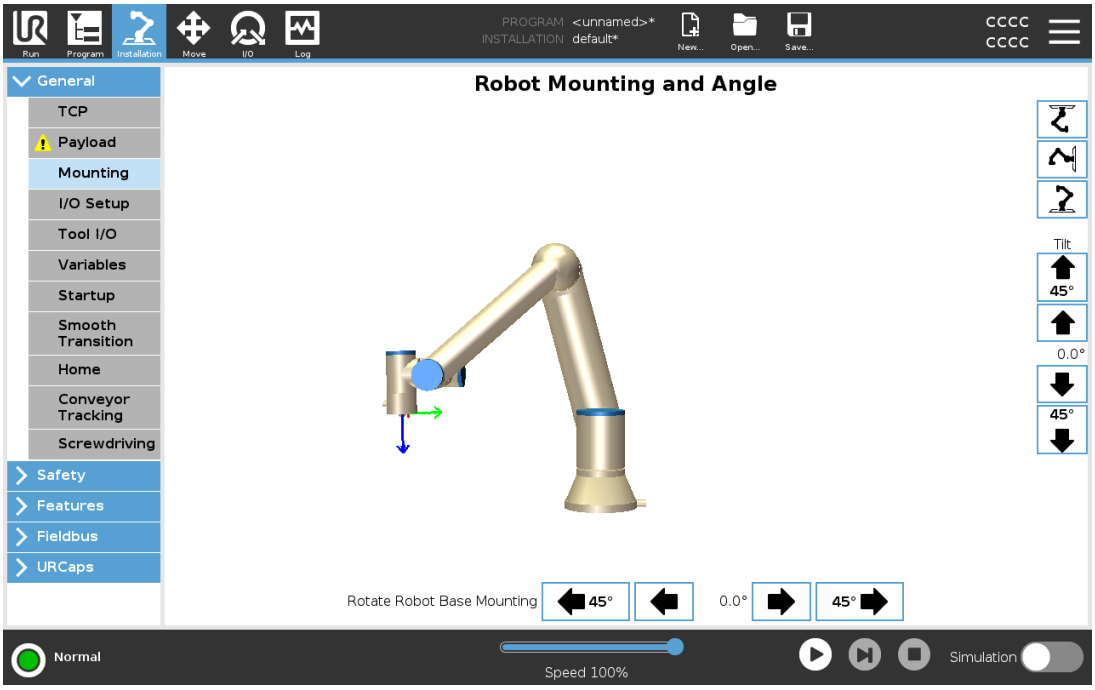
Failure to mount the Robot's arm correctly may result in frequent robot stops, and/or the Robot arm will move when pressing the **Freedrive** button.

If the Robot arm is mounted on a flat table or floor, no change is needed on this screen. However, if the Robot arm is **ceiling mounted**, **wall mounted**, or **mounted at an angle**, this needs to be adjusted using the buttons.
The buttons on the right side of the screen are for setting the angle of the Robot arm's mounting. The top three right side buttons set the angle to **ceiling** (180°), **wall** (90°), **floor** (0°). The **Tilt** buttons set an arbitrary angle.

The buttons on the lower part of the screen are used to rotate the mounting of the Robot arm to match the actual mounting.

WARNING

Use the correct installation settings. Save and load the installation files with the program.



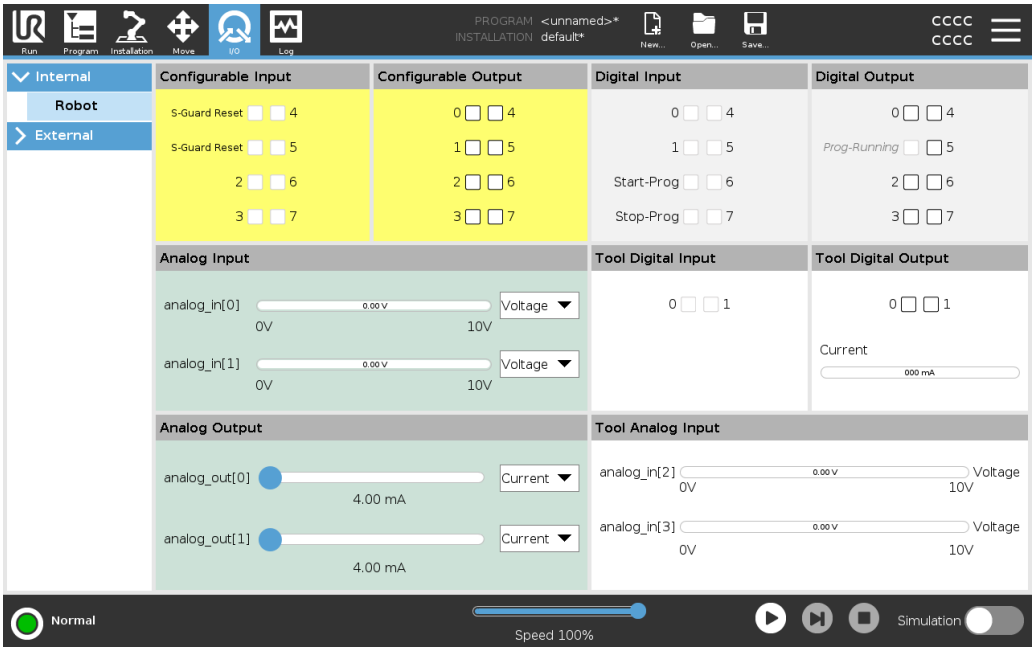
10.5.12. Using the I/O Tab

Description

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

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

Configurable I/Os can be reserved for special safety settings defined in the safety I/O configuration section of the installation (see I/O); those which are reserved will have the name of the safety function in place of the default or user defined name.
Configurable outputs that are reserved for safety settings are not togglable and will be displayed as LED's only.



Voltage

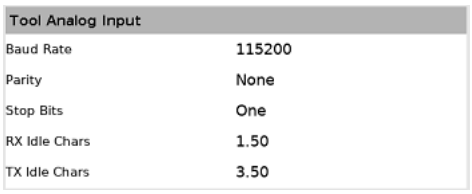
When the Tool Output is controlled by the user, you can configure Voltage. Selecting a URCap removes access to Voltage.

Analog Domain Settings

The analog I/O's can be set to either current [4-20mA] or voltage [0-10V] output. These settings are persistent over restarts of the robot controller and saved in the installation. Control over the tool I/Os could be assigned to a URCap in **Tool I/O** of the **Installation** tab. Selecting a URCap removes user's control over tool's analog I/O.

Tool Communication Interface

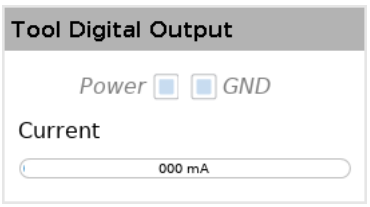
When the **Tool Communication Interface TCI** is enabled, the tool analog input becomes unavailable. On the **I/O** screen, the **Tool Input** field appears as shown.



Dual Pin power

When **Dual Pin Power** is enabled, the tool digital outputs must be named as follows:

- tool_out[0] (Power)
- tool_out[1] (GND)



10.5.13. Analog Input: Communication Interface

Description

The Tool Communication Interface (TCI) enables the robot to communicate with an attached tool via the robot tool analog input. This removes the need for external cabling. Once the Tool Communication Interface is enabled, all tool analog inputs are unavailable

Tool Communication Interface

1. Tap the Installation tab and under General tap Tool I/O.
2. Select Communication Interface to edit TCI settings.
   Once the TCI is enabled, the tool analog input is unavailable for the I/O Setup of the Installation and does not appear in the input list. Tool analog input is also unavailable for programs as Wait For options and expressions.
3. In the drop-down menus under Communication Interface, select required values. Any changes in values are immediately sent to the tool. If any installation values differ from what the tool is using, a warning appears.

10.5.14. Digital Output

Description

The tool communication interface allows two digital outputs to be independently configured. In PolyScope, each pin has a drop-down menu that allows the output mode to be set. The following options are available:

- Sinking: This allows the pin to be configured in an NPN or Sinking configuration. When the output is off, the pin allows a current to flow to the ground. This can be used in conjunction with the PWR pin to create a full circuit.
- Sourcing: This allows the pin to be configured in a PNP or Sourcing configuration. When the output is on, the pin provides a positive voltage source (configurable in the IO Tab). This can be used in conjunction with the GND pin to create a full circuit.
- Push / Pull: This allows the pin to be configured in a Push / Pull configuration. When the output is on, the pin provides a positive voltage source (configurable in IO Tab). This can be used in conjunction with the GND pin to create a full circuit When the output is off, the pin allows a current to flow to the ground.

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

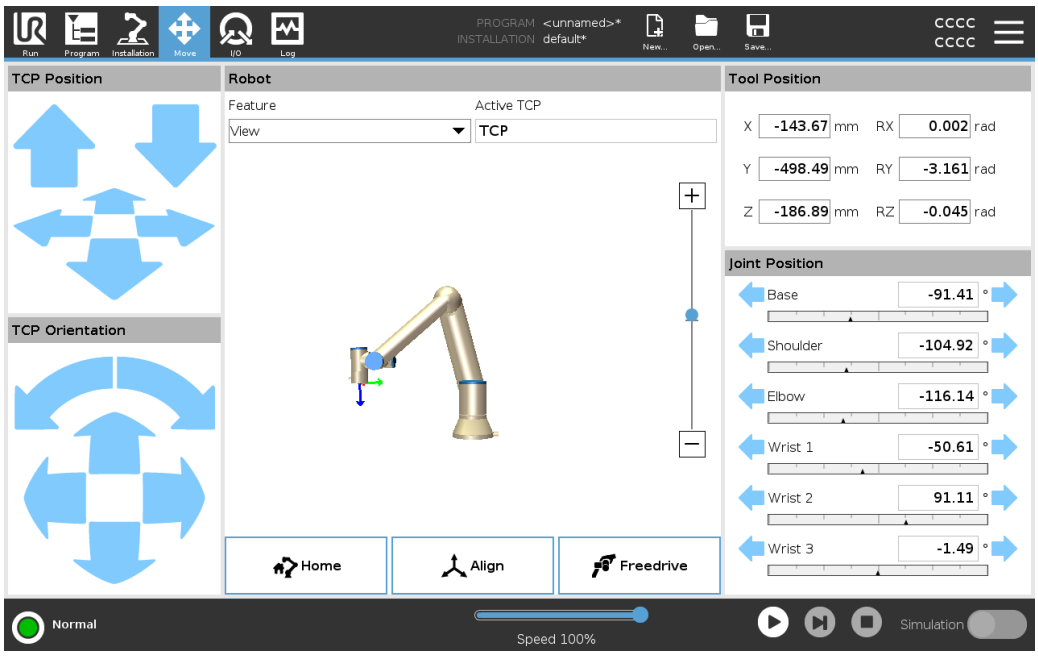
Dual Pin Power

Dual Pin Power is used as a source of power for the tool. Enabling Dual Pin Power disables default tool digital outputs.

10.5.15. Using the Move Tab

Description

Use the Move Tab screen to move (jog) the robot arm directly, either by translating/rotating the robot tool, or by moving robot joints individually.



To use the Move Tool arrows

Hold down any of the **Move Tool** arrows to move the robot arm in the corresponding direction.

- The **Translate arrows** (upper) move the tool flange in the direction indicated.
- The **Rotate arrows** (lower) change the orientation of the tool in the indicated direction. The rotation point is the Tool Center Point (TCP), i.e.the point at the end of the robot arm that gives a characteristic point on the tool. The TCP is shown as a small blue ball.

Robot

If the current position of the TCP approaches a safety plane, a trigger plane, or the orientation of robot tool is near the tool orientation boundary limit , a 3D representation of the proximate boundary limit is shown. The visualization of boundary limits is disabled during program execution.

Safety planes display in yellow and black with an arrow indicating which side of the plane, the robot TCP is allowed to be positioned.

Trigger planes display in blue and green with an arrow indicating the side of the plane, where the **Normal** mode limits are active.

The tool orientation boundary limit is visualized with a spherical cone together with a vector indicating the current orientation of the robot tool. The inside of the cone represents the allowed area for the tool orientation (vector).
When the robot TCP is no longer in proximity of the limit, the 3D representation disappears. If the TCP is in violation or very close to violating a boundary limit, the visualization of the limit turns red.

Feature

Under **Feature**, you can define how to control the robot arm relative to **View**, **Base** or **Tool** features. For the best feel for controlling the robot arm you can select the **View** feature, then use **Rotate arrows** to change the viewing angle of the 3D image to match your view of the real robot arm.

Active TCP

In the **Robot** field, under **Active TCP**, the name of the current active Tool Center Point (TCP) is displayed.

Home

The **Home** button accesses the **Move Robot into Position** screen, where you can hold down the **Auto** button to move robot into position previously defined under Installation. The Home button's default setting returns the Robo Arm to an upright position .

Freedrive

The on-screen **Freedrive** button allows the Robot Arm to be pulled into desired positions/poses.

Align

The **Align** button allows the Z axis of the active TCP to align to a selected feature.

Tool Position The text boxes display the full coordinate values of the TCP relative to the selected feature. You can configure several named TCPs (see). You can also tap **Edit pose** to access the **Pose Editor** screen.

Joint Position The **Joint Position** field allows you to directly control individual joints. Each joint moves along a default joint limit range from − 360° to + 360°, defined by a horizontal bar. Once the limit is reached you cannot move a joint any further. You can configure joints with a position range different from the default, this new range is indicated with red zone inside the horizontal bar.

Using Freedrive in the Move tab The **Freedrive** button shall only be used in applications if allowed by the risk assessment.

WARNING

Failure to correctly configure the mounting setting can result in unwanted robot arm movement when you use the **Freedrive** button.

- Payload settings and robot mounting settings shall be set correctly before using Freedrive.
- All personnel shall remain outside the reach of the robot arm, when **Freedrive** is in use.

WARNING

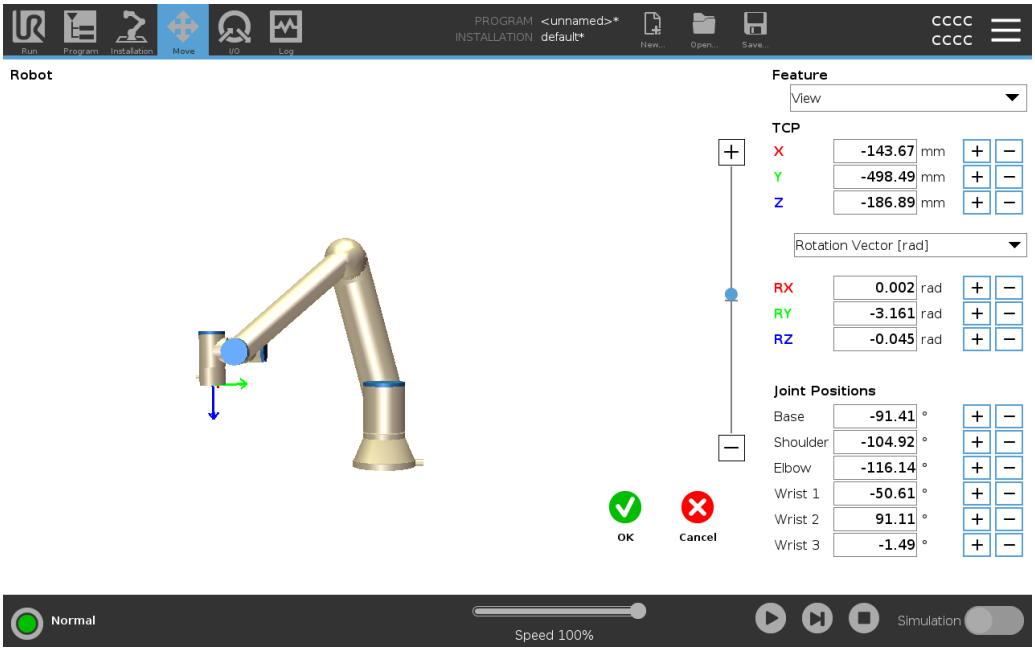
Failure to correctly configure the installation settings, can increase the risk of the robot arm falling during **Freedrive**, due to payload errors.

- Verify the installation settings are correct (e.g. Robot mounting angle, payload mass and payload center of gravity offset) . Save and load the installation files along with the program.
- Save and load the installation files along with the program.

10.5.16. Pose Editor

Description

Once you access the **Pose Editor** screen, you can precisely configure a target joint positions, or a target pose (position and orientation) for the TCP. Note: This screen is **offline** and does not control the Robot Arm directly.



Robot

The 3D image shows the current Robot Arm position. The **shadow** shows the Robot Arm target position controlled by the specified values on the screen. Press the magnifying glass icons to zoom in/out or drag a finger across it to change the view.
If the specified target position of the robot TCP is close to a safety or trigger plane, or the orientation of robot tool is near the tool orientation boundary limit, a 3D representation of the proximate boundary limit is shown. Safety planes are visualized in yellow and black with a small arrow representing the plane normal, which indicates the side of the plane on which the robot TCP is allowed to be positioned. Trigger planes are displayed in blue and green and a small arrow pointing to the side of the plane, where the **Normal** mode limits are active. The tool orientation boundary limit is visualized with a spherical cone together with a vector indicating the current orientation of the robot tool. The inside of the cone represents the allowed area for the tool orientation (vector). When the target robot TCP is no longer in proximity of the limit, the 3D representation disappears. If the target TCP is in violation or very close to violating a boundary limit, the visualization of the limit turns red.

Feature and Tool Position

The active TCP and coordinate values of the selected feature are displayed. The **X**, **Y**, **Z** coordinates specify tool position. The **RX**, **RY**, **RZ** coordinates specify orientation.
Use the drop down menu above the **RX**, **RY** and **RZ** boxes to choose the orientation representation type:

- **Rotation Vector** `[rad]` The orientation is given as a *rotation vector*. The length of the axis is the angle to be rotated in radians, and the vector itself gives the axis about which to rotate. This is the default setting.
- **Rotation Vector** `[°]` The orientation is given as a *rotation vector*, where the length of the vector is the angle to be rotated in degrees.
- **RPY** `[rad]` *Roll*, *pitch* and *yaw* (*RPY*) angles, where the angles are in radians. The RPY-rotation matrix (X, Y', Z" rotation) is given by:
  $R_{rpy}(\gamma, \beta, \alpha) = R_Z(\alpha) \cdot R_Y(\beta) \cdot R_X(\gamma)$
- **RPY** `[°]` *Roll*, *pitch* and *yaw* (*RPY*) angles, where angles are in degrees.

You can tap the values to edit the coordinates. You can also tap the **+** or **-** buttons to the right of a box to add/subtract an amount to/from the current value. Or you can hold down a button to directly increase/decrease the value.

Joint Positions

Individual joint positions are specified directly. Each joint position can have Joint Limit range from − 360° to + 360°. You can configure Joint Positions as follows:

- Tap the joint position to edit the values.
- Tap the **+** or **-** buttons to the right of a box to add or subtract an amount to/from the current value.
- Hold down a button to directly increase/decrease the value.

OK Button

If you activate this screen from the **Move** screen (see), tap the **OK** button to return to the **Move** screen. The Robot Arm moves to the specified target. If the last specified value was a tool coordinate, the Robot Arm moves to the target position using movement type **MoveL**; or it uses movement type **MoveJ** if a joint position was specified last.

Cancel Button

The **Cancel** button exits the screen discarding all changes.