Operating manual

RobotStudio

5.12

Document ID: 3HAC032104-001

Revision: B

ABB AB
Robotics Products
721 68 Västerås
Sweden

Overview

About This Manual

This manual describes how to create, program and simulate robot cells and stations using RobotStudio. For online programming, this manual describes how to supervise, install, configure and program a real robot controller. Terms and concepts related to offline and online programming are also explained.

RobotStudio offers the following installation options:

- Full
- Custom, allowing user-customized contents and paths
- Minimal, running RobotStudio in Online mode only

Usage

This manual should be used when working with either the offline or online functions of RobotStudio.

Who Should Read This Manual?

This manual is intended for RobotStudio users, proposal engineers, mechanical designers, offline programmers, robot technicians and service technicians.

Prerequisites

The reader should have basic knowledge of:

- Robot programming
- Generic Windows handling
- 3D CAD programs

Organization of Chapters

The manual is organized into six main chapters containing procedures and explanations.

These are followed by descriptions of the commands of the Graphical User Interface (GUI), loosely arranged—in order to be read from the application as online help files—according to menu or tab.

Chapter	Contents
1. Introduction	Contains installation instructions, basic explanations of the terms and concepts related to robotics and programming, and a description of the GUI.
2. How to build stations	Describes how to build stations in RobotStudio. This includes importing and configuring the equipment to be simulated, as well as testing the reachability for finding the optimal station layout.
3. How to program robots	Describes how to create robot movements, I/O signals, process instructions and logics in a RAPID program for the robots. It also describes how to run and test the program.
4. How to simulate programs	Describes how to simulate and validate robot programs.

Continues on next page

Chapter	Contents
5. Deployment and distribution	Describes how to transfer systems between RobotStudio's virtual controllers and real IRC5 controllers, how to copy programs, how to package an active station for moving between RobotStudio PCs, and how to capture a screen.
6. Working online	Covers the functionality of the Minimal Installation, describing such online functions as building systems (with offline examples), handling I/O and events, and configuring systems.

References

Reference	Document Id
Product manual - IRC5	3HAC021313-001
Operating manual - IRC5 with FlexPendant	3HAC16590-1
Technical reference manual - RAPID overview	3HAC16580-1
Technical reference manual - System parameters	3HAC17076-1
Application manual - MultiMove	3HAC021272-001
Application manual - Conveyor tracking	3HAC16587-1

Revisions

Revision	Description
A	First revision, called RobotStudio 2008, released for Partner Days. Tne entire manual has been adapted to the new GUI, in which RobotStudio$^{On-line}$ has been integrated.
B	Updated the manual with information on Conveyor Tracking, Operator Window, Station Viewer, Linked Geometry, Viewpoints and Recorder simulation.

Product documentation, M2004

Categories for robot documentation

The robot documentation is divided into a number of categories. This listing is based on the type of information in the documents, regardless of whether the products are standard or optional.

All documents listed can be ordered from ABB on a DVD. The documents listed are valid for M2004 robot systems.

Product manuals

All hardware, robots and controllers, will be delivered with a **Product manual** that contains:

- Safety information.
- Installation and commissioning (descriptions of mechanical installation, electrical connections).
- Maintenance (descriptions of all required preventive maintenance procedures including intervals).
- Repair (descriptions of all recommended repair procedures including spare parts).
- Additional procedures, if any (calibration, decommissioning).
- Reference information (article numbers for documentation referred to in Product manual, procedures, lists of tools, safety standards).
- Part list.
- Foldouts or exploded views.
- Circuit diagrams.

Technical reference manuals

The technical reference manuals describe the robot software in general and contain relevant reference information.

- **RAPID Overview**: An overview of the RAPID programming language.
- **RAPID Instructions, Functions and Data types**: Description and syntax for all RAPID instructions, functions, and data types.
- **RAPID Kernel**: A formal description of the RAPID programming language.
- **System parameters**: Description of system parameters and configuration workflows.

Application manuals

Specific applications (for example software or hardware options) are described in **Application manuals**. An application manual can describe one or several applications.

An application manual generally contains information about:

- The purpose of the application (what it does and when it is useful).
- What is included (for example cables, I/O boards, RAPID instructions, system parameters, CD with PC software).
- How to use the application.
- Examples of how to use the application.

Continues on next page

Operating manuals

The operating manuals describe hands-on handling of the products. The manuals are aimed at those having first hand operational contact with the product, that is production cell operators, programmers, and trouble shooters.

The group of manuals includes (among others):

- **Emergency safety information**
- **General safety information**
- **Getting started, IRC5 and RobotStudio**
- **IRC5 with FlexPendant**
- **RobotStudio**
- **Introduction to RAPID**
- **Trouble shooting**, for the controller and robot.

Safety

Safety of personnel

A robot is heavy and extremely powerful regardless of its speed. A pause or long stop in movement can be followed by a fast hazardous movement. Even if a pattern of movement is predicted, a change in operation can be triggered by an external signal resulting in an unexpected movement.

Therefore, it is important that all safety regulations are followed when entering safeguarded space.

Safety regulations

Before beginning work with the robot, make sure you are familiar with the safety regulations described in the manual *General safety information*.

1 Introduction

1.1 Terms and Concepts

1.1.1. Hardware concepts

Overview

This section introduces the hardware in a typical IRC5 robot cell. For detailed explanations, see the manuals related to IRC5 robots specified in *References on page 11*.

Standard hardware

The table below describes the standard hardware in an IRC5 robot cell.

Hardware	Explanation
Robot manipulator	An ABB industrial robot.
Control module	Contains the main computer that controls the motion of the manipulator. This includes RAPID execution and signal handling. One control module can be connected to 1 – 4 drive modules.
Drive module	A module containing the electronics that power the motors of a manipulator. The drive module can contain up to nine drive units, each controlling one manipulator joint. Since the standard robot manipulators have six joints, you usually use one drive module per robot manipulator.
FlexController	The controller cabinet for the IRC5 robots. It consists of one control module and one drive module for each robot manipulator in the system.
FlexPendant	The programming pendant, connected to the control module. Programming on the FlexPendant is referred to as “online programming”.
Tool	A device usually mounted on the robot manipulator to allow it to perform specific tasks, such as gripping, cutting or welding. The tool can also be stationary, see below for more information.

Optional hardware

The table below describes the optional hardware for an IRC5 robot cell.

Hardware	Explanation
Track manipulator	A moving stand holding the robot manipulator to give it a larger work space. When the control module controls the motion of a track manipulator, it is referred to as a “Track External Axis”.
Positioner manipulator	A moving stand normally holding a work piece or a fixture. When the control module controls the motion of a positioner manipulator, it is referred to as an “External Axis”.
FlexPositioner	A second robot manipulator acting as a positioner manipulator. It is controlled by the same control module as the positioner manipulator.
Stationary tool	A device that stands in a fixed location. The robot manipulator picks up the work piece and brings it to the device to perform specific tasks, such as gluing, grinding or welding.
Work piece	The product being worked on.

Continues on next page

Continued

Hardware	Explanation
Fixture	A construction holding the work piece in a specific position so that the repeatability of the production can be maintained.

1.1.2. RobotWare concepts

Overview

This section introduces terminology regarding RobotWare. For detailed explanations, see the manuals related to IRC5 robots specified in *References on page 11*.

RobotWare

The table below describes the RobotWare terminology and concepts that can be useful when working with RobotStudio.

Concept	Explanation
RobotWare	As a concept, refers to both the software used to create a RobotWare System and the RobotWare systems themselves.
RobotWare DVD	Delivered with each control module. On the DVD you will find the RobotWare installation and some other useful software. Check the Release Notes on your DVD for specifications.
RobotWare installation	When installing RobotWare on a PC, you install into the mediapool the specific versions of the files from which RobotStudio uses to create the RobotWare system. When installing RobotStudio, only one version of RobotWare will be installed. To simulate a specific RobotWare system, the RobotWare version used for this particular RobotWare system must be installed on your PC.
RobotWare Key	Used when you create a new RobotWare system or upgrade an existing system. The RobotWare keys unlock the RobotWare options included in the system, and determine the RobotWare version from which the RobotWare system will be built. For IRC5 systems there are three types of RobotWare keys: • The controller key, which specifies the controller and software options. • The drive keys, which specify the robots in the system. The system has one drive key for each robot it uses. • Additional option keys, which specify additional options, like positioner external axes. A virtual key allows you to select any RobotWare options you wish, but a RobotWare system created from a virtual key can only be used in a virtual environment such as RobotStudio.
RobotWare system	A set of software files that, when loaded into a controller, enables all functions, configurations, data and programs controlling the robot system. RobotWare systems are created in the RobotStudio software. The systems can be stored and saved on a PC, as well as on the control module. RobotWare systems can be edited by RobotStudio or the FlexPendant.
RobotWare version	Each RobotWare is released with a major and a minor version number, separated by a dot. The RobotWare version for IRC5 is 5.xx, where xx identifies the minor version. When ABB releases a new robot model, a new RobotWare version will be released with support for the new robot.

Continues on next page

Continued

Concept	Explanation
Mediapool	The mediapool is a folder on the PC in which each RobotWare version is stored in a folder of its own. The files of the mediapool are used to create and implement all the different RobotWare options. Therefore, the correct RobotWare version must be installed in the mediapool when creating RobotWare systems or running them on virtual controllers.

1.1.3. RAPID concepts

Overview

This section introduces the basic terminology of RAPID. For detailed explanations, see the manuals related to RAPID and programming specified in *References on page 11*.

Terminology of the RAPID structure

The table below describes the RAPID terminology that you may come across when working with RobotStudio. The concepts are listed by size, from most basic to increasingly large.

Concept	Explanation
Data declaration	Used to create instances of variables or data types, like num or tooldata.
Instruction	The actual code commands that make something happen, for example, setting data to a specific value or a robot motion. Instructions can only be created inside a routine.
Move instructions	Create the robot motions. They consist of a reference to a target specified in a data declaration along with parameters that set motion and process behavior. If inline targets are used, the position is declared in the move instructions.
Action instruction	Instructions that perform other actions than moving the robot, such as setting data or sync properties.
Routine	Usually a set of data declarations followed by a set of instructions implementing a task. Routines can be divided into three categories: procedures, functions and trap routines.
Procedure	A set of instructions that does not return a value.
Function	A set of instructions that returns a value.
Trap	A set of instructions that is triggered by an interrupt.
Module	A set of data declarations followed by a set of routines. Modules can be saved, loaded and copied as files. Modules are divided into program modules and system modules.
Program module (.mod)	Can be loaded and unloaded during execution.
System module (.sys)	Used mainly for common system-specific data and routines, for example, an arcware system module that is common for all arc robots.
Program files (.pgf)	In IRC5 a RAPID program is a collection of module files (.mod) and the program file (.pgf.) that references all the module files. When loading a program file, all old program modules are replaced by those referenced in the .pgf file. System modules are unaffected by program load.

1.1.4. Concepts of programming

Overview

This section introduces the terminology regarding programming. For detailed explanations, see the manuals related to programming and IRC5 Robots specified in *References on page 11*.

Programming concepts

The table below describes the terminology and concepts that are used in robot programming.

Concept	Explanation
Online programming	Programming connected to the control module. This expression also implies using the robot to create positions and motion.
Offline programming	Programming without being connected to the robot or the control module.
True offline programming	Refers to the ABB Robotics concept of connecting a simulation environment to a virtual controller. This enables not only program creation, but also program testing and optimizing offline.
Virtual controller	A software that emulates a FlexController to allow the same software (the RobotWare system) that is controlling the robots to run on a PC. This gives the same behavior of the robots offline as you get online.
MultiMove	Running multiple robot manipulators with the same control module.
Coordinate systems	Used to define positions and orientations. When programming a robot, you can take advantage of using different coordinate systems to more easily position objects relative to each other.
Frame	A synonym for coordinate system.
Workobject calibration	If all your targets refer to workobjects, you only need to calibrate the workobjects when deploying offline programs.

1.1.5. Targets and paths

Overview

Targets (positions) and paths (sequences of move instructions to targets) are used when programming robot motions in RobotStudio.

When you synchronize the RobotStudio station to the virtual controller, RAPID programs are created from the paths.

Targets

A target is a coordinate that the robot shall reach. It contains the following information:

Information	Description
Position	The position of the target, defined in a workobject coordinate system, see *Coordinate systems on page 22*.
Orientation	The orientation of the target, relative to the orientation of the workobject. When the robot reaches the target, it will align the TCP's orientation with the target's orientation, see *Coordinate systems on page 22*.
Configuration	Configuration values that specify how the robot shall reach the target. For more information, see *Robot axis configurations on page 24*.

Targets are converted to instances of the data type *robtarget* when synchronized to the virtual controller.

Paths

A sequence of move instructions, paths are used to make the robot move along a sequence of targets.

Paths are converted to procedures when synchronized to the virtual controller.

Move instructions

A move instruction consists of:

- a reference to a target
- motion data, such as motion type, speed and zone
- a reference to a tooldata
- a workobject reference

Action instructions

An action instruction is a RAPID string that can be used for setting and changing parameters. Action instructions can be inserted before, after or between instruction targets in paths.

1.1.6. Coordinate systems

Overview

This section gives a short introduction to coordinate systems that are often used for offline programming. In RobotStudio, you can use all coordinate systems described below, as well as user-defined coordinate systems, for relating objects and elements to each other.

Hierarchy

The coordinate systems relate to each other hierarchically, where the origin of each coordinate system is defined as a position in one of its ancestries. Below are descriptions of commonly used coordinate systems, starting at the top of the hierarchy.

World coordinate system

The world coordinate system represents the entire station or robot cell. This is the top of the hierarchy to which all other coordinate systems are related.

Task frame coordinate system

The task frame coordinate system is useful to define within a station or robot cell, in particular with multiple robots or mechanisms. One task frame can coordinate the placement of several mechanisms, whereas several task frames are suitable when working with MultiMove Independent.

Base coordinate system

Each robot in the station has a base coordinate system, which is always located at the base of the robot.

Tool Center Point coordinate system

The tool center point coordinate system, also called TCP, is the center point of the tool. Several different TCPs may be defined for one robot. All robots have one predefined TCP at the robot's tool mounting point, called *tool0*.

When a program runs, the robot moves the TCP to the programed position.

Continues on next page

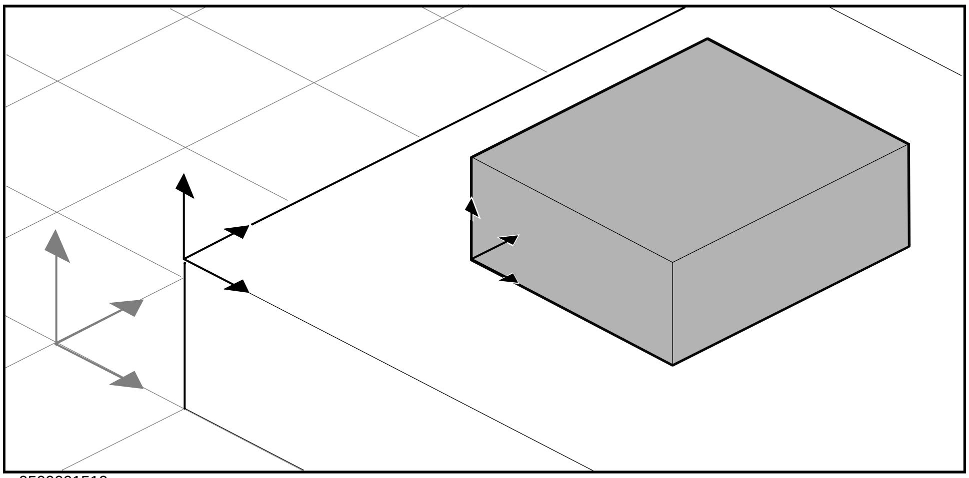
WorkObject coordinate system

The workobject normally represents the physical work piece. It is composed of two coordinate systems: the *User frame* and the *Object frame*, where the latter is a child to the former. When programming a robot, all targets (positions) are related to the object frame of a workobject. If no other workobject is specified, the targets will be related to the default *Wobj0*, which always coincides with the base frame of the robot.

Using workobjects provides the chance to easily adjust robot programs with an offset, if the location of the work piece has been changed. Thus, workobjects can be used for calibrating offline programs. If the placement of the fixture/work piece relative to the robot in the real station does not completely match the placement in the offline station, you simply adjust the position of the workobject.

Workobjects are also used for coordinated motions. If a workobject is attached to a mechanical unit (and the system uses the option for coordinated motions), the robot will find the targets in the workobject even when the mechanical unit moves the workobject.

In the picture below the grey coordinate system is the world coordinate system, and the black ones are the object frame and the user frame of the workobject. Here the user frame is positioned at the table/fixture and the object frame at the workpiece.



xx0500001519

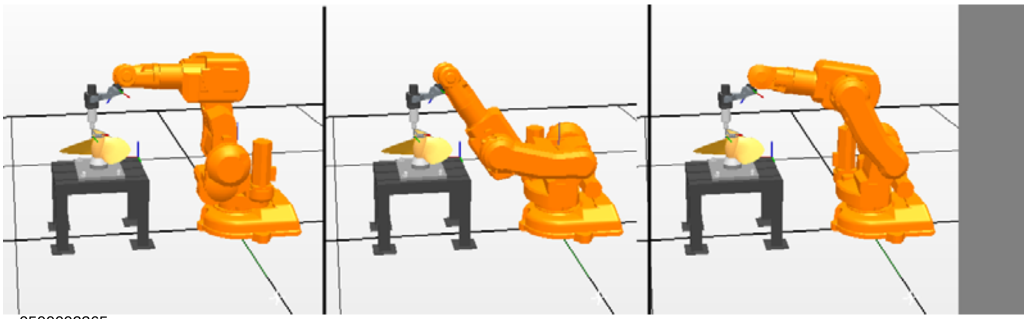
User Coordinate Systems

User Coordinate Systems (UCSs) are used for creating reference points of your choice. For example, you can create UCSs at strategic points in the work piece to facilitate programming.

1.1.7. Robot axis configurations

Axis configurations

Targets are defined and stored as coordinates in a WorkObject coordinate system. When the controller calculates the position of the robot axes for reaching the target, it will often find more than one possible solution to configuring the robot axes.



xx0500002365

To distinguish between the different configurations, all targets have a configuration value that specifies the quadrant in which each axis shall be located.

Storing axis configurations in targets

For targets that are taught after jogging the robot to the position, the used configuration will be stored in the target.

Targets created by specifying or calculating positions and orientations get a default configuration value (0,0,0,0), which might not be valid for reaching the target.

Common problems related to robot axis configurations

It is most likely that targets created by other ways than jogging cannot be reached at their default configuration.

Even if all targets in a path have validated configurations, you might encounter problems when running the path if the robot cannot move from one configuration to the other. This is likely to occur where an axis shifts greater than 90 degrees during linear movements.

Repositioned targets keep their configuration, but the configurations are no longer validated. As a result, the problems described above might occur when moving targets.

Common solutions for configuration problems

To resolve the problems described above, you can assign a valid configuration to each target and verify that the robot can move along each path. You can also turn configuration monitoring off, which means that you ignore the stored configurations and let the robot find working configurations at runtime. If this is not done the proper way, you might get unexpected results.

In some cases there might not be any working configurations. Possible solutions might then be to reposition the work piece, reorient targets (if acceptable for the process) or add an external axis that either moves the work piece or the robot for increasing reachability.

Continues on next page

How configurations are denoted

The robot's axis configurations are denoted by a series of four integers, specifying in which quadrant of a full revolution significant axes are located. The quadrants are numbered from zero for positive (counterclockwise) rotation and from -1 for negative (clockwise) rotation.

For a linear axis, the integer specifies the range (in meters) from the neutral position in which the axis is located.

A configuration for a six-axis industrial robot (like IRB 140) may look like:

```
[0 -1 2 1]
```

The first integer (0) specifies the position of axis 1: somewhere in the first positive quadrant (between 0 and 90 degrees rotation).

The second integer (-1) specifies the position of axis 4: somewhere in the first negative quadrant (between 0 and -90 degrees rotation).

The third integer (2) specifies the position of axis 6: somewhere in the third positive quadrant (between 180 and 270 degrees rotation).

The fourth integer (1) specifies the position of axis x, a virtual axis used for specifying the wrist center in relation to other axes.

Configuration monitoring

When executing a robot program, you can choose whether to monitor configuration values. If configuration monitoring is turned off, configuration values stored with the targets are ignored, and the robot will use the configuration closest its current configuration for reaching the target. If turned on, it will only use the specified configuration for reaching the targets.

Configuration monitoring can be turned off and on for joint and linear movements independently and is controlled by the ConfJ and ConfL action instructions.

Turning configuration monitoring off

Running a program without configuration monitoring may result in different configurations each time a cycle is executed: When the robot returns to the start position after completing a cycle, it may choose a different configuration then the original.

For programs with linear move instructions this might cause a situation where the robot gets closer and closer its joint limits and eventually will not be able to reach the target.

For programs with joint move instructions this might cause sweeping, unpredictable movements.

Turning configuration monitoring on

Running a program with configuration monitoring forces the robot to use the configurations stored with the targets. This results in predictable cycles and predictable motions. In some situations, however, like when the robot moves to a target from an unknown position, using configuration monitoring may limit the robot's reachability.

When programming offline, you must assign a configuration to each target if the program shall be executed with configuration monitoring.

1.1.8. Libraries, geometries and CAD files

Overview

For programming or simulating in RobotStudio, you need models of your work pieces and equipment. Models for some standard equipment are installed as libraries or geometries with RobotStudio. If you have CAD models of your work pieces and custom equipment, these can be imported as geometries to RobotStudio. If you do not have CAD models, you can create them in RobotStudio.

Difference between geometries and libraries

The objects you import to a station can be either geometries or libraries.

Geometries are basically CAD files, which, when imported, are copied to the RobotStudio station.

Libraries are objects that have been saved in RobotStudio as external files. When you import a library, a link from the station to the library file is created. Accordingly, the station file does not grow in the same way as when importing geometries. Furthermore, besides the geometrical data, library files can contain RobotStudio-specific data. For example, if a tool is saved as a library, the tool data is saved together with the CAD data.

How geometries are constructed

An imported geometry is displayed as one part in the Objects browser. From RobotStudio's Modeling tab, you can see the components of the geometry.

The top node of the geometry is called a **Part**. The part contains **Bodies**, which can be of the types solid, surface or curve.

Solid bodies are 3D objects, made up of **Faces**. You recognize a true 3D solid by this one body containing multiple faces.

Surface bodies are 2D objects of just one face. A part that contains several bodies with one face each that together constitute a 3D object is created from 2D surfaces, and is therefore not a true 3D solid. If these parts are not created correctly, they might cause problems both in their display and graphical programming. see *Troubleshooting and optimizing geometries on page 71*.

Curved bodies, represented by the body node alone in the Modeling browser, do not contain any child nodes.

From the Modeling tab, you can edit the parts by adding, moving, rearranging or deleting bodies. Thus, you can optimize existing parts by removing unnecessary bodies, as well as create new parts by grouping bodies.

Importing and converting CAD files

For importing geometries from single CAD files, you use RobotStudio's import function, see *Importing a station component on page 68*.

If you need to convert CAD files to other formats or want to change the default settings for the conversion before making the import, you can use the CAD converter installed with RobotStudio before making the import, see *Converting CAD formats on page 70*.

Continues on next page

Supported CAD formats

Some of the CAD formats require a separate license for being imported or converted by RobotStudio. The table below shows the supported CAD formats and whether a license is required:

Format	File extensions	License requirement	Default target formats
Acis, reads/writes versions v6 to R18	sat	No	Iges, Step, Vdafs
Iges, reads to version 5.3, writes version 5.3	igs, iges	Yes	Acis, Step, Vdafs
Step, reads versions AP203 and AP214 (geometry only), writes version AP 214	stp, step, p21	Yes	Acis, Step, Vdafs
Vdafs, reads to 2.0, writes 2.0	vda, vdafs	Yes	Acis, Iges, Step
Catia V4, reads versions 4.1.9 to 4.2.4	model, exp	Yes	Acis, Iges, Step, Vdafs
Catia V5, reads versions R2 - R18	CATPart, CATProduct	Yes	Acis, Iges, Step, Vdafs
Pro/Engineer, reads versions 16 to Wildfire3	prt, asm	Yes	Acis, Iges, Step, Vdafs
Inventor, reads versions 6 to 12	ipt	Yes	Acis, Iges, Step, Vdafs
Vrml	wrl, vrml,vrml1, vrml2	No	RsGfx
Jupiter, up to 6.4	jt	No	RsGfx
STL	stl	No	RsGfx
PLY	ply	No	RsGfx
3DStudio	3ds	No	RsGfx

You need licenses for both the source format and the target format when converting licensed formats.

Continues on next page

Mathematical versus graphical geometries

A geometry in a CAD file always has an underlying mathematical representation. Its graphical representation, displayed in the graphics window, is generated from the mathematical representation when the geometry is imported to RobotStudio, after which the geometry is referred to as a part.

For this kind of geometry, you can set the detail level of the graphical representation, thus reducing the file size and rendering time for large models and improving the visual display for small models you might want to zoom in on. The detail level only affects the visual display; paths and curves created from the model will be accurate both with coarse and fine settings.

A part can also be imported from a file that simply defines its graphical representation; in this case, there is no underlying mathematical representation. Some of the functions in RobotStudio,such as snap mode and creation of curves from the geometry, will not work with this kind of part.

To customize the detail level settings, see *RobotStudio Options on page 178*.

1.1.9. VSTA as the IDE

Overview

RobotStudio uses Microsoft Visual Studio Tools for Applications (VSTA) as its Integrated Development Environment (IDE), enabling advanced users to extend and customize its functionality. You can, for example, write an add-in in C# or VB.Net to create a toolbar or macro, debug code or inspect variable values during execution.

In addition, the Add-In browser acts as a single window for both VSTA add-ins and non-VSTA, RobotStudio-specific add-ins known as PowerPacs.

For a procedure, see *Visual Studio Tools for Applications on page 318*.

Types of add-ins

The following add-ins may be available in the Add-In browser:

Object	Description
PowerPac	An add-in that accords with RobotStudio specifications, but is not part of the VSTA system itself.
VSTA Station Add-In	A VSTA add-in that has been attached to a station, and saved to the station file.
VSTA User Add-In	A VSTA add-in that has not been attached to a station and is active only on the current user's computer.

Using the shortcut menus of the Add-In browser, PowerPacs may be loaded or autoloaded, while VSTA add-ins may be added, loaded, unloaded, reloaded, autoloaded, edited, deleted, or removed from the station.

1.2. Installing and Licensing RobotStudio

Installing RobotStudio

RobotStudio offers the following installation options:

- **Minimal** - Installs only program features required to run RobotStudio in Online mode
- **Complete** - Installs all program features required to run the complete RobotStudio
- **Custom** - Installs user -customized program features

How to install RobotStudio on a PC

	Action
1.	Insert the robot software DVD in the PC. If a menu for the DVD is opened automatically, continue with step 5. If no menu for the DVD is opened, continue with step 2.
2.	On the **Start** menu, click **Run**.
3.	In the **Open** box, type the drive letter for your DVD drive followed by: `:\launch.exe` Example: If your DVD drive has the letter D, then type: `D:\launch.exe`
4.	Click **OK**.
5.	Select language for the DVD menu.
6.	On the DVD menu, click **Install**.
7.	On the installation menu, click **RobotStudio**. This opens the installation wizard, which will guide you through the rest of the software installation.
8.	Follow the instructions in the installation wizard.
9.	After installing RobotStudio, proceed with activating RobotStudio. **NOTE!** For a trial period of 30 days, RobotStudio will work without activation.

Activate RobotStudio

To continue using the product with all of its features, you must activate it. RobotStudio Product Activation is based on Microsoft anti-piracy technology and designed to verify that software products are legitimately licensed.

Activation works by verifying that the Activation Key is not in use on more personal computers than are permitted by the software license.

How do I activate RobotStudio?

When you start RobotStudio for the first time after installation, you are prompted to enter your 25-digit Activation Key (xxxxx-xxxxx-xxxxx-xxxxx-xxxxx).

Trial period

Before entering a valid Activation Key, you can run the software, in Premium functionality mode, with all the features enabled, for a trial period of up to 30 days.

NOTE! Trial period starts immediately after installation.

After entering a valid Activation Key, you will see only the features you have purchased.

NOTE ! If installed during the trial period, you will lose the trial period.

Continues on next page

Basic Functionality mode

After the trial period, the software reverts to Reduced Functionality mode unless you have entered a valid Activation Key. In Basic functionality mode, RobotStudio allows only the use of the Online and basic Virtual Controller features. No existing files or stations are harmed in Reduced Functionality mode. After activating your software, you will have full functionality for the features you have purchased.

NOTE!

Activation is not required for the Online features for programming, configuring and monitoring a real controller connected over Ethernet.

Activate automatically over the Internet or manually

The Activation Wizard gives you two choices on how to proceed.

Automatic activation by using the Internet (recommended)

Once you have selected the option *Activate RobotStudio over the Internet*, the Activation Wizard automatically contacts the ABB licensing servers over your Internet connection. If you are using a valid Activation Key that has not exceeded the number of installations allowed, your product is activated immediately.

When you activate over the Internet, your activation request is sent to ABB. Your license will then be automatically installed and your product ready for use. If you choose to activate over the Internet but are not currently connected, the wizard alerts you that there is no connection.

Manual activation

If the computer does not have an Internet connection, you must create a license file by selecting the option *Create a license request file*. Proceed through the wizard, enter your Activation Key and save the License Request File to your computer. Use a removable medium, such as a USB stick or floppy disk, to transfer the file to a computer with an Internet connection. Go to *www.robotstudio.com/community* to find a hyperlink to the manual activation page, click the link and then follow the instructions. The result will be a License File that should be saved and transferred back to the computer holding your product. Relaunch the Activation Wizard and select the option *Install a license file*. Proceed through the wizard, selecting the License File when requested. Upon completion, RobotStudio is activated and ready for use.

How do I activate later?

If you do not want to activate your copy of the software at installation, you can do so later. The following steps will launch the Activation Wizard.

If you have a problem with your activation, contact your local ABB customer support representative at the e-mail address or telephone number provided at *www.abb.com/robotics*.

	Action
1.	Click **RobotStudio Button**, then click **RobotStudio Options** button and select **Licensing**.
2.	Click **Activation wizard** to launch the activation wizard.
3.	Proceed through the wizard to complete the activation. **NOTE!** If the RobotStudio installation is activated, you will have valid licenses for the features covered by your subscription.

Continues on next page

Which RobotStudio version is installed?

The version number of RobotStudio is displayed on the start page that appears when RobotStudio is started.

How can I tell if my RobotStudio installation is activated?

	Action
1.	Click **RobotStudio Button**, then click **RobotStudio Options** button, and select **Licensing**.
2.	Click **View Installed License Keys** to see the status of your current license.
3.	If the RobotStudio installation is activated, you will have valid licenses for the features covered by your subscription.

1.3 The Graphical User Interface

1.3.1. The Getting Started window

Overview

The Getting Started window contains the following tabs:

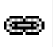
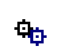
Tab	Description
Recent Stations Offline	Use this tab to open recent stations (which are previewed), a new station or any saved station.
Recent Controllers Online	Use this tab to open a recent controller, connect to a controller with one click or add a controller.
Information	Use this tab to access help files, browse tutorials, connect to the online community, manage your licenses or browse RobotStudio news.

1.3.2. The Layout browser

Overview

The layout browser is a hierarchical display of physical items, such as robots and tools.

Icons

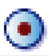
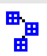
Icon	Node	Description
 xx0500001364	**Robot**	The robot in the station. The red lock in the lower right corner of the icon indicates that the object is connected to a library.
 xx0500001365	**Tool**	A tool.
 xx0500001366	**Link collection**	Contains all the links of the objects.
 xx0500001367	**Link**	A physical object in a joint connection. Each link is made up of one or several parts.
 xx0500001478	**Frames**	Contains all the frames for an object.
 xx0500001368	**Component group**	A grouping of parts or other assemblies, carrying its own coordinate systems. It is used to structure a station.
 xx0500001369	**Part**	A physical object in RobotStudio. Parts with geometric information are made up of one or more 2D or 3D entities. Parts without geometric information (such as imported .jt files) are empty.
 xx0500001370	**Collision set**	Contains all collision sets. Each collision set includes two groups of objects.
 xx0500001371	**Objects group**	Contains references to the objects that are subject to collision detection.
 xx0500001479	**Collision set mechanisms**	The objects in the collision set.
 xx0500001372	**Frame**	The frames in the station.

1.3.3. The Paths & Targets browser

Overview

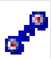
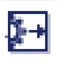
The paths & targets browser is a hierarchical display of non-physical items.

Icons

Icon	Node	Description
 xx0500001373	**Station**	Your station in RobotStudio.
 xx0500001374	**Virtual Controller**	The system for controlling the robots, just like a real IRC5 controller.
 xx0500001375	**Task**	Contains all logical elements in the station, such as targets, paths, workobjects, tooldata and instructions.
 xx0500001376	**Tooldata Collection**	Contains all tooldata.
 xx0500001471	**Tooldata**	A tooldata for a robot or a task.
 xx0500001377	**Workobjects & Targets**	Contains all workobjects and targets for the task or robot.
 xx0500001477	**Jointtarget Collection and Jointtarget**	A specified position of the robot axes.
 xx0500001378	**Workobject Collection and Workobject**	The workobject collection node and the workobjects it contains.
 xx0500001379	**Target**	A defined position and rotation for a robot. A target equals a RobTarget in a RAPID program.
 xx0500001849	Target without assigned configuration	A target for which no axis configuration has been assigned, for example, a repositioned target or a new target created by means other than teaching.
 xx0500001850	Target without found configuration	An unreachable target, that is, for which no axis configuration has been found.
 xx0500001380	**Path Collection**	Contains all paths in the station.

Continues on next page

Continued

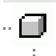
Icon	Node	Description
 xx0500001381	**Path**	Contains the instructions for the robot movements.
 xx0500001474	**Linear Move Instruction**	A linear TCP motion to a target. If the target has no valid configuration assigned, the move instruction gets the same warning symbols as the target.
 xx0500001851	**Joint Move Instruction**	A joint motion to a target. If the target has no valid configuration assigned, the move instruction gets the same warning symbols as the target.
 xx0500001475	**Action Instruction**	Defines an action for the robot to perform at a specified location in a path.

1.3.4. The Modeling browser

Overview

The modeling browser is a display of editable objects and their building blocks.

Icons

Icon	Node	Description
 xx0600002704	**Part**	Geometric items corresponding to the objects in the **Layout** browser.
 xx0600002705	**Body**	Geometric building blocks that comprise the parts. 3D bodies contain several faces, 2D bodies one face, and curves no faces.
 xx0600002706	**Face**	The faces of the bodies.

1.3.5. The Offline and Online browsers

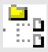
Overview

The offline and online browsers is a hierarchical display of controller and configuration elements.

Icons

Icon	Node	Description
 xx0300000026	**Controllers**	Contains the controllers that are connected to the Robot View.
 xx0300000027	**Connected Controller**	Represents a controller with a working connection.
 xx0400000677	**Connecting Controller**	Represents a controller which is currently being connected.
 xx0300000028	**Disconnected Controller**	Represents a controller that has lost its connection. It might have been turned off or disconnected from the network.
 xx0600003287	**Denied login**	Represent a controller that denies you access to login. Possible reasons for denied access: • The user lacks grant to login • Too many clients connected to the controller • The RobotWare version of the system running on the controller is newer than the version of RobotStudio Online
 xx0300000029	**Configuration**	Contains the configuration topics.
 xx0300000030	**Topic**	Each parameter topic is represented by a node: • Communication • Controller • I/O • Man-machine communication • Motion
 xx0300000037	**Event Log**	With the Event Log you can view and save controller events.
 xx0300000434	**I/O System**	Represents the controller I/O system. The I/O system consists of I/O buses and units.

Continues on next page

 xx0300000433	**I/O Bus**	An I/O bus is a connector for one or several I/O units.
 xx0300000432	**I/O Unit**	An I/O Unit is a board, panel or any other device with ports through which the I/O signals are sent.
 xx0300000031	**RAPID Tasks**	Contains the active tasks (programs) of the controller.
 xx0300000032	**Task**	A task is a robot program, which executes alone or together with other programs. A program is composed of a set of modules.
 xx0300000431	**Modules**	A container for either program modules or system modules.
 xx0300000430	**Program Module**	A program module contains a set of data declarations and routines for a specific task. Program modules contains data specific for this program.
 xx0300000428	**System Module**	A system module contains a set of type definitions, data declarations and routines. System modules contains data that applies to the robot system, regardless which program modules that are loaded.
 xx	**Encrypted Module**	A module that is locked for editing and viewing.
 xx	**Nostepin Module**	A module that cannot be entered during step-by-step execution. That is all instructions in the module are treated as one if the program is executed step-by-step.
 xx0300000429	**Procedure**	A routine that does not return any value. Procedures are used as subprograms.
 xx0300000034	**Function**	A routine that returns a value of a specific type.
 xx0300000036	**Trap**	A routine that provides a means of responding to interrupts.
 xx0300000039	**Documents Folder**	Folder for links to documents and files.

Continues on next page

Continued

 xx0300000038	**Document Link**	Link to a document. The icon for the document is the same as the Window icon for the document type.

1.3.6. The Output window

Overview

The output window displays information about events that occur in the station, such as when simulations are started or stopped. This information is useful when troubleshooting stations.

Layout of the Output tab

The **Output** tab contains two columns: the first states the event, the second the time the message was generated. Each row is a message.

Event types

The three event types indicate the severity of the event:

Event type	Description
Information	An information message is a normal system event, for example, starting and stopping programs, changing the operational mode, and turning motors on and off. Information messages never require an action from you. They can be useful for tracking errors, collecting statistics or monitoring user-triggered event routines.
Warning	A warning is an event of which you should be aware, but it is not so severe that the process or RAPID program needs to be stopped. Warnings must occasionally be acknowledged. Warnings often indicate underlying problems that at some point will need to be resolved.
Error	An error is an event that prevents the robot system from proceeding. The running process or RAPID program cannot continue and will be stopped. An error must occasionally be acknowledged. Some errors require some immediate action from you in order to be resolved. Double-click an error to display a detailed information box.

Some of the events are active. These are linked to an action for resolving the problem that generated the event. To activate the linked action, double-click the message.

Handling messages in the Output window

Goal	Procedure
To filter messages...	Right-click in the Output window and then click Show messages. From the options **All Errors**, **information**, **Warnings**and **Warnings and Errors**, select the type of messages you want to display.
To save a message to file...	Select it, right-click and then click Save to file. Choose a name and location in the dialog box. Multiple messages can be selected by pressing SHIFT while clicking each.
To clear the Output window...	Right-click in the Output window and then click **Clear**.

Continues on next page

The Controller tab

The controller status window shows the operational status of the controllers in your robot view.

1. The **System Name** column

   This column shows the name of the system that is running on the controller.

2. The **Controller Name** column

   This column shows the name of the controller.

3. The **Controller State** column

   This column shows the state of the controller:

When the controller is in state...	the robot is...
Initializing	starting up. It will shift to state *motors off* when it has started.
Motors off	in a standby state where there is no power to the robot's motors. The state has to be shifted to motors on before the robot can move.
Motors on	ready to move, either by jogging or by running programs.
Guard Stop	stopped because the safety runchain is opened. For instance, a door to the robot's cell might be open.
Emergency Stop	stopped because emergency stop was activated.
Waiting for motors on after e-stop	ready to leave emergency stop state. The emergency stop is no longer activated, but the state transition isn't yet confirmed.
System Failure	in a system failure state. A warm start is required.

4. The **Program Execution State** column

   This column shows if the robot is running any program or not:

When the controller is in state...	the robot...
Running	is running a program.
Ready	has a program loaded and is ready to run it when a PP (starting point in the program) has been set.
Stopped	has a program loaded, with a PP, and is ready to run it.
Uninitialized	has not initialized the program memory. This indicates an error condition.

5. The **Operating Mode** column

   This column shows the operating mode of the controller:

When the controller is in mode...	the robot is...
Initializing	starting up. It will shift to the mode selected on the controllers cabinet when it has started.
Auto	ready to run programs in production. In Auto mode it is possible to get remote Write access to the controller, which is necessary for editing programs, configurations and other things with RobotStudio Online.

Continues on next page

When the controller is in mode...	the robot is...
Manual	only able to move if the enabling device on the FlexPendant is activated. Furthermore, the robot can only moved with reduced speed in manual mode. In manual mode it is not possible to get remote Write access to the controller, unless it is configured for this and the remote Write access granted on the FlexPendant.
Manual full speed	only able to move if the enabling device on the FlexPendant is activated. In manual mode it is not possible to get remote Write access to the controller, unless it is configured for this and the remote Write access granted on the FlexPendant.
Waiting for acknowledge	about to enter Auto mode, but the mode transition has not yet been acknowledged.

6. The **Logged on as** column

   This column shows the user name the PC is logged on to the controller with.

7. The **Access** column

   This column shows who has write access to the controller, or if it is available.

8. The **Connection Type** column

   This column shows the type of connection to the controller.

1.3.7. Operator Window

Overview

Operator Window is an alternative to the corresponding feature in the Virtual FlexPendant for communicating with the user during RAPID program execution. It displays the same output as displayed on the Virtual FlexPendant Operator Window.

When running in a Virtual Controller, the RAPID program communicates with the operator via messages on the FlexPendant screen. The Operator Window integrates this functionality and allows the user to run interactive RAPID programs without starting the Virtual FlexPendant.

Enabling Operator Window

To enable an operator window:

1. On the **Application menu,** click **RobotStudio Options.**
2. On the **Navigation pane** to the left, select **Robotics:Virtual Controller**.
3. On the **Virtual Controller** page to the right, select **Show virtual Operator Window** option.
4. Click **Apply**.

NOTE!

When the **Show virtual Operator Window** feature is enabled, an Operator Window is automatically created for each controller in the station. By default, the window is located in the tab area below the graphics window.

RAPID Instructions

List of RAPID instructions supported by the Operator Window:

- TPErase
- TPReadFK
- TPReadNum
- TPWrite
- UIAlphaEntry
- UIMsgBox
- UINumEntry

NOTE! When these instructions are executed, the behavior is similar to that of Virtual FlexPendant.

List of RAPID instructions not supported by the Operator Window:

- TPShow
- UIShow
- UINumTune
- UIListView

NOTE! When these instructions are executed, an error message is displayed in the Operator Window prompting you to use the Virtual Flexpendant instead.

Continues on next page

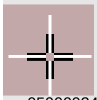
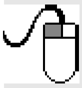
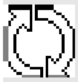
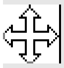
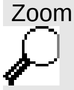
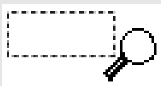
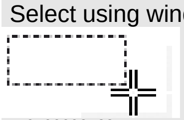
NOTE!

You should not run both the Virtual Flexpendant and Operator Window simultaneously.

1.3.8. Using a mouse

Navigating the graphics window using the mouse

The table below shows how to navigate the graphics window using the mouse:

To	Use the keyboard / mouse combination	Description
Select items  xx0500002417	 xx0500002421	Just click the item to select. To select multiple items, press CTRL key while clicking new items.
Rotate the station  xx0500002424	**CTRL** + **SHIFT** +  xx0500002421	Press CTRL + SHIFT + the left mouse button while dragging the mouse to rotate the station. With a 3-button mouse you can use the middle and right buttons, instead of the keyboard combination.
Pan the station  xx0500002422	**CTRL** +  xx0500002421	Press CTRL + the left mouse button while dragging the mouse to pan the station.
Zoom the station  xx0500002426	**CTRL** +  xx0500002423	Press CTRL + the right mouse button while dragging the mouse to the left to zoom out. Dragging to the right zooms in. With a 3-button mouse you can also use the middle button, instead of the keyboard combination.
Zoom using window  xx0500002425	**SHIFT** +  xx0500002423	Press SHIFT + the right mouse button while dragging the mouse across the area to zoom into.
Select using window  xx0500002428	**SHIFT** +  xx0500002421	Press SHIFT + the left mouse button while dragging the mouse across the area to select all items that match the current selection level.

1.3.9. Selecting an item

Overview

Each item in a station can be moved to achieve the required layout, so you first have to determine its selection level. The selection level makes it possible to select only specific types of items, or specified parts of objects.

The selection levels are curve, surface, entity, part, mechanism, group, target/frame and path. The target/frame and path selection can be combined with any of the other selection levels.

Objects may also be grouped together as component groups, see *Component Group on page 226*.

Selecting an item in the graphics window

To select items in the graphics window, follow these steps:

1. At the top of the graphics window, click the desired selection level icon.
2. Optionally, click the desired snap mode icon for the part of the item you wish to select.
3. In the graphics window, click the item. The selected item will be highlighted.

Multiple selection of items in the graphics window

To select multiple items in the graphics window, do the following:

1. Press the **SHIFT** key, and in the graphics window drag the mouse diagonally over the objects to select.

Selecting an item in the browsers

To select items in a browser, do the following:

1. Click the item. The selected item will be highlighted in the browser.

Multiple selection of items in the browsers

To select multiple items in a browser, follow these steps:

1. Make sure that all the items to be selected are of the same type and located in the same branch of the hierarchical structure; otherwise, the items will not be operable.
2. Do one of the following:
   - To select adjacent items: In the browser, hold down the **SHIFT** key and click the first and then the last item. The list of items will be highlighted.
   - To select separate items: In the browser, hold down the **CTRL** key and click the items you want to select. The items will be highlighted.

1.3.10. Attaching and detaching objects

Overview

You can attach an object (child) to another object (parent). Attachments can be created on part level and on mechanism level. When an object has been attached to a parent, moving the parent also moves the child.

One of the most common attachments is to attach a tool to a robot. For procedures, see *Attach to on page 323* and *Detach on page 331*.

1.3.11. Keyboard Shortcuts

Keyboard Shortcuts

The following table lists the keyboard shortcuts.:

Command	Key Combination
General	
Activate menu bar	F10
Open API Help	ALT + F1
Open Help	F1
Open Virtual FlexPendant	CTRL + F5
Switch between windows	CTRL + TAB
General Commands	
Add Controller System	F4
Open Station	CTRL + O
Take Screenshot	CTRL + B
Teach Move Instruction	CTRL + SHIFT + R
Teach Target	CTRL + R
Toggle View Representation	F7
General Editing	
Copy	CTRL + C
Cut	CTRL + X
Paste	CTRL + V
Redo	CTRL + Y
Refresh	F5
Rename	F2
Select All	CTRL + A
Undo	CTRL + Z
Import Geometry	CTRL + G
Import Library	CTRL + J
New Station	CTRL + N
Save Station	CTRL + S
Program Editor Intellisense	
Complete Word	CTRL + SPACEBAR
Parameter Info	CTRL + I
PickList	CTRL + SHIFT + SPACEBAR
Program Editor Commands	
Start Program Execution	F8
Step In	F11
Step Out	SHIFT + F11
Step Over	F12
Stop	SHIFT + F8

Continues on next page

Command	Key Combination
Toggle Breakpoint	F9
Apply Changes	CTRL + SHIFT + S
Print	CTRL + P
Program Editor Hotkeys	
Copy	CTRL + Insert
Cut	SHIFT + Delete
Cut marked lines	CTRL + L
Delete marked lines	CTRL + SHIFT + L
Delete to beginning of word	CTRL + BACKSPACE
Delete to end of word	CTRL + Delete
Indent	Tab
Make the selected text lowercase	CTRL + U
Make the selected text uppercase	CTRL + SHIFT + U
Move to beginning of document	CTRL + Home
Move to beginning of line	Home
Move to end of document	CTRL + End
Move to end of line	End
Move to next word	CTRL + Right
Move to previous word	CTRL + Left
Move to visible bottom	CTRL + Page Down
Move to visible top	CTRL + Page Up
Open line above	CTRL + Enter
Open line below	CTRL + SHIFT + Enter
Outdent	SHIFT + TAB
Paste	SHIFT + Insert
Redo	CTRL + SHIFT + Z
Scroll down	CTRL + Down
Scroll up	CTRL + Up
Select block down	ALT + SHIFT + Down
Select block left	ALT + SHIFT + Left
Select block right	ALT + SHIFT + Right
Select block to next word	CTRL + ALT + SHIFT + Right
Select block to previous word	CTRL + ALT + SHIFT + Left
Select block up	ALT + SHIFT + Up
Select down	SHIFT + Down
Select left	SHIFT + Left
Select page down	SHIFT + Page Down
Select page up	SHIFT + Page Up
Select right	SHIFT + Right
Select to beginning of document	CTRL + SHIFT + Home
Select to beginning of line	SHIFT + Home
Select to end of document	CTRL + SHIFT + End

Continues on next page

Command	Key Combination
Select to end of line	SHIFT + End
Select to next word	CTRL + SHIFT + Right
Select to previous word	CTRL + SHIFT + Left
Select to visible bottom	CTRL + SHIFT + Page Down
Select to visible top	CTRL + SHIFT + Page Up
Select up	SHIFT + Up
Select word	CTRL + SHIFT + W
Toggle overwrite mode	Insert
Transpose characters	CTRL + T
Transpose lines	CTRL + ALT + SHIFT + T
Transpose words	CTRL + SHIFT + T

2 How to build stations

2.1. Workflow for building a new station

Overview

This is an overview of how to build a station by including the equipment necessary to create and simulate robot programs.

The first part of this overview describes the alternatives for creating a station with a system. The second part describes importing or creating the objects to work with. The third part describes the workflow for optimizing the station layout by finding the best placement of robots and other equipment.

In most cases, following the workflows from start to finish is recommended, even if other sequences are also possible.

Creating a station with a system

The table below shows the alternatives for creating a station with a system.

For procedures, see *New Station on page 172*.

Activity	Description
Create a station with a template system	This is the simplest way to create a new station containing a robot and a link to a rudimentary system template.
Create a station with an existing system	This creates a new station containing one or more robots in accordance with an existing, built system.
Create a station with no system	An advanced user can build a station from scratch and then add a new or existing system to it.

Manually starting the VC

The table below shows the alternatives for manually starting with a system. Perform only those steps applicable to your station.

Activity	Description
Manually connecting a library to the VC	See *Starting a VC on page 64*.
Restarting the VC	See *Restarting a VC on page 66*.

Importing station components

The table below shows the workflow for importing station components. Perform only those steps applicable to your station.

For procedures, see *Importing a station component on page 68*.

Activity	Description
Import a robot model	See *Robot System on page 188*.
Import a tool	See *Import Library on page 187*.
Import a positioner	See *ABB Library on page 186*.
Import a track	See *Import Library on page 187*.
Import other equipment	If you have CAD models of the equipment, you can import them, see *Import Library on page 187*. Otherwise, you can create models in RobotStudio, see *Mechanisms on page 75*.

Continues on next page

Activity	Description
Add work piece	If you have CAD models of the work piece, you can import them, see *Workobject on page 195*. Otherwise, you can create models in RobotStudio, see *Objects on page 73*.

Placing objects and mechanisms

The table below shows the workflow for placing the objects in the station.

Activity	Description
Place objects	If you are building a model of a real station, start by placing all objects with known positions. For objects without known positions, find a suitable placement, see *Placing objects* and *Placing external axes on page 79*.
Attach tools	Attach the tools to the robot, see *Attach to on page 323*.
Attach robots to tracks	If track external axes are used, attach the robots to the tracks, see *Attach to on page 323*.
Attach work pieces to positioners	If positioner external axes are used, attach the work pieces to the positioners, see *Attach to on page 323*.
Test reachability	Test if the robot can reach critical positions on the work piece. If you are satisfied with how the robot reaches the positions, your station is ready for programming. Otherwise, continue adjusting the placement or trying other equipment as described below, see *Testing positions and motions on page 100*.

2.2 Setting up a conveyor tracking station with two robots working on the same conveyor

2.2.1. Two robot systems sharing the same task frame position

Overview

This section describes what happens when both the robot systems share the same task frame position. The baseframes of the mechanical units in both the robot systems have the same task frame position.

Prerequisite

- Two robot systems with conveyor tracking option (system 1 and system 2)
- A conveyor mechanism saved as library

NOTE! See *Create Conveyor mechanism on page 247* to create conveyor tracking systems.

Setting up the conveyor tracking station

1. Add the existing system (system 1) to the station. See *Robot System on page 188*.

   NOTE! After starting the system, when asked to select the library, browse and select the already saved conveyor mechanism library.
2. Modify the baseframe positions of conveyor and robot.
   1. Move the mechanical unit (conveyor/robot) to its new location.
   2. See *Updating the baseframe position on page 294* to update the baseframe position of the conveyor/robot.
   3. Repeat steps 1 and 2to modify the baseframe position of the robot.
   4. In the **System Configuration** window, click **OK**. When asked if you want to restart the system, answer **Yes.** Close the **System Configuration** window.
3. Add the existing system (system 2) to the station. See *Robot System on page 188*.

   NOTE! After starting the system, when asked to select the library, browse and select the same library as the one selected for system 1 or any other library. Later, this conveyor library will be removed from the station since system 2 shall use the same conveyor library as system 1.
4. Refer both systems (system 1 and system 2) to the same conveyor library.
   1. In the **Offline** browser, click **System Configuration** to bring up a dialog box for system 2.
   2. Select the library node in the hierarchical tree.
   3. Select the option **Select from Station**. Click **Change**. The **Select Library** dialog box appears.
   4. Select the same conveyor library as the one selected for system 1. Click **OK**.

Continues on next page

NOTE! Now both systems (system 1 and system 2) use the same conveyor library and the library previously referenced by system 2 is removed from the station.

5. Modify the baseframe positions of robot (system 2).

1. Move the mechanical unit (robot) to its new location.
2. See *Updating the baseframe position on page 294*to update the baseframe position of the robot.
3. Repeat steps 1 and 2 to modify the baseframe position of the robot.
4. In the **System Configuration** window, click **OK**. When asked if you want to restart the system, answer **Yes.** Close the **System Configuration** window.

2.2.2. Two robot systems having different task frame positions

Overview

This section describes what happens when two robot systems have different task frame positions but uses the same sync switch. This means the Baseframes of the conveyor mechanical units in both the robot systems have different values.

Prerequisites

Two robot systems with conveyor tracking option (system 1 and system 2)

NOTE! See *Create Conveyor mechanism on page 247* to create conveyor tracking systems.

Setting up the conveyor tracking station

1. Add the existing system (system 1) to the station. See *Robot System on page 188*.

   NOTE! After starting the system, when asked to select the library, browse and select the already saved conveyor mechanism library.
2. Modify the baseframe positions of conveyor and robot.
   1. Move the mechanical unit (conveyor/robot) to its new location.
   2. See *Updating the baseframe position on page 294* to update the baseframe position of the conveyor/robot.
   3. Repeat steps 1 and 2 to modify the baseframe position of the robot.
   4. In the **System Configuration** window, click **OK**. When asked if you want to restart the system, answer **Yes.** Close the **System Configuration** window.
3. Add the existing system (system 2) to the station. See *Robot System on page 188*.

   NOTE! After starting the system, when asked to select the library, browse and select the same library as the one selected for system 1 or any other library. Later, this conveyor library will be removed from the station since system 2 shall use the same conveyor library as system 1.
4. Update both systems (system 1 and system 2) to use the same conveyor library.
   1. In the **Offline** browser, click **System Configuration** to bring up a dialog box for system 2.
   2. Select the library node in the hierarchical tree.
   3. Select the option **Select from Station**. Click **Change**. The **Select Library** dialog box appears.
   4. Select the same conveyor library as the one selected for system 1. Click **OK**.

   NOTE! Now both systems (system 1 and system 2) use the same conveyor library and the library previously referenced by system 2 is removed from the station.
5. Modify the task frame position of the conveyor mechanism. See *Set Task Frames on page 293*.

   NOTE! Before modifying the task frame, make a note of the current conveyor position in world coordinates. After modifying the task frame, move the conveyor back to the position it was before modifying the task frame.

Continues on next page

6. Modify the baseframe positions of robot (system 2).

Repeat step 2 to modify the baseframe position of the robot (system 2)

1. Move the mechanical unit (robot) to its new location.
2. See *Updating the baseframe position on page 294* to update the baseframe position of the robot.
3. Repeat steps 1 and 2 to modify the baseframe position of the robot.
4. In the **System Configuration** window, click **OK**. When asked if you want to restart the system, answer **Yes.** Close the **System Configuration** window.

7. Modify the baseframe position of the conveyor (system 2).

1. In the **Offline** browser, click **System Configuration** to bring up a dialog box for system 2.
2. Select the conveyor in the hierarchical tree. The BaseFrame property list for the conveyor is now displayed.
3. Select the option **Use Current Station Values** to update the baseframe value of the robot in the controller.
4. Deselect the option **Check BaseFrame on Startup**.
5. In the **System Configuration** window, click **OK**. When asked if you want to restart the system, answer **Yes**.

NOTE! By deselecting the option **Check BaseFrame on Startup**, RobotStudio will not compare the BaseFrame values in the station and the controller every time the controller is started. This avoids repositioning the conveyor library.

NOTE! If the two robot systems use the same part on the conveyor, the relation between the part and the two conveyor workobjects should be the same.

2.3. Creating a system with external axes automatically

Automatically create a system with external axes

1. Create a robot system from layout. See *Robot System on page 188*.
2. Import the desired robots, positioners, and track libraries while running through the wizard. See *Import Library on page 187*.

   NOTE! If a robot and track are selected, attach the robot to the track. See *Attach to on page 323*.

   NOTE! Robot system supports the following tracks with lengths 1.7 m to 19.7 m in a separate task or same robot task. Depending on the manipulator type, the system allows one to three tracks per task. However with IRBTx004, only one track of this type can be used per system.

   - IRBT4003
   - IRBT4004
   - IRBT6003
   - IRBT6004
   - IRBT7003
   - IRBT7004
   - RTT_Bobin
   - RTT_Marathon
   - Paint Rail

Supported external axes configuration

The following table shows a combination of different external axes configurations:

Combination	Positioner type							
	A	B	C	D	K	L	2xL	R
One IRB (Positioner in same task)	Y	Y	Y	Y	Y	Y	Y	Y
One IRB (Positioner in separate task)	Y	Y	Y	Y	Y	Y	Y	Y
Two IRB (Positioner in separate task)	Y	Y	Y	Y	Y	Y	N	Y
One IRB on Track Motion (Positioner in same task)	Y	N	N	N	Y	YX	Y	N
One IRB on Track Motion (Positioner in separate task)	Y	N	N	N	Y	YX	Y	N
Two IRB on Track Motion (Positioner in separate task)	Y	N	N	N	Y	YX	N	N

- Y - Combination is supported
- N - Combination is not supported
- YX - Combination is supported and manual mapping of mechanical units and joints required

Continues on next page

Manual mapping of mechanical units and joints

If the system contains more than one mechanical unit, the number of tasks and base frame positions of the mechanism should be verified in the System Configuration.

1. In the **Offline** browser, click **System Configuration** to bring up a dialog box.
2. Select the robot from the node in the hierarchical tree.

   The property page of this node contains controls for mapping and setting axes and joints.
3. Click **Change** to open a dialog box.
4. Manually map the mechanical unit and mechanism joints. Click **Apply**.
5. Modify the baseframe positions of the mechanical unit. See *Updating the baseframe position on page 294*.

2.4 Manually setting up a system with track motion

2.4.1. Setting up a system with track motion of type RTT or IRBTx003 manually

Manually set up a system with track motion of type RTT or IRBTx003

Use this procedure to manually set up a system with track motion type RTT Bobin, RTT Marathon or IRBT4003, IRBT6003, or IRBT7003.

1. Build and start a new system. See *Building a new system on page 142*.

	Action	Description
1	Select the desired robot variant (IRB6600).	On the **Modify Options** page of the **System Builder**, scroll down to **Drive Module 1 > Drive module application** group and expand **ABB Standard manipulator** option and select **Manipulator type** (IRB6600).
2	Select **Additional axes configuration**.	On the **Modify Options** page of the **System Builder**, scroll down to **Drive Module 1> Additional axes configuratin** group and expand the **Add axes IRB/drive module 6600** option and select the **770-4 Drive W in pos Y2** option. **NOTE!** The option **770-4 Drive W in pos Y2**, the **Drive module**, and the **Position** varies depending on the **Additional axes configuration** selected. Make sure to select at least one drive in any position.
3	Click **Finish**.	Close the **Modify Options** page.

2. Add the system to the station. See *Adding a system on page 65*

3. Add the corresponding track configuration file of the desired robot variant (IRB 6600) and the desired track model to the station. See *Adding the track to the system on page 68*.

NOTE! In the **Select Library** group, select either the existing track or import a different track

NOTE! The system may fail unless the correct additional axes configuration is selected.

4. Specify whether the baseframe is moved by another mechanism.

1. In the **Offline** browser, click **System Configuration** to bring up a dialog box.
2. Select **ROB_1** node from the hierarchical tree.
3. Select the option **Track** from the **BaseFrame moved by** list.
4. Click **OK**. When asked if you want to restart the system, answer **Yes**. Close the **System Configuration** window.

2.4.2. Setting up a system with track motion of type IRBTx004 manually

Overview

For configuration of tracks IRBT4004, IRBT6004 or IRBT7004, the additional option mediapools are installed in the same Mediapool folder as that of RobotWare. Search path ....\\ProgramFiles\ABB Industrial IT\Robotics IT\MediaPool\.

The following three Track mediapool versions are installed in RobotStudio:

- Track.5.09.0012, supports RobotWare 5.09
- Track.5.10.0003 and Track.5.10.0005, supports RobotWare 5.10
- Track.5.11.0001, supports RobotWare 5.11

Manually set up a system with track motion of type IRBTx004

1. Build and start a new system. See *Building a new system on page 142*.

	Action	Description
1	Add additional options for IRBTx004.	See *Adding additional options on page 143*. **NOTE!** Browse and select the key file (.kxt) located in the mediapool Track 5.11.0001.
2	Select the desired robot variant (IRB6600).	On the **Modify Options** page of the **System Builder**, scroll down to **Drive Module 1 > Drive module application** group and expand **ABB Standard manipulator** option and select **Manipulator type** (IRB6600).
3	Select **Additional axes configuration**.	On the **Modify Options** page of the **System Builder**, scroll down to **Drive Module 1> Additional axes configuratin** group and expand the **Add axes IRB/drive module 6600** option and select the **770-4 Drive W in pos Y2** option. **NOTE!** The option **770-4 Drive W in pos Y2**, the **Drive module**, and the **Position** varies depending on the **Additional axes configuration** selected. Make sure to select at least one drive in any position.
4	Select the desired track motion (IRBT 6004).	On the **Modify Options** page of the **System Builder**, scroll down to the **TRACK** and expand the **Drive module for Track motion** group. Select **Drive Module 1 >Track Motion type > IRBT 6004 > Irb Orientation on Track > Standard carriage In Line > Select Track Motion Length > 1.7m (or any other variant).**
5	Click **Finish**.	Close the **Modify Options** page.

2. Add the system to the station. See *Adding a system on page 65*.

3. Add the corresponding track configuration file of the desired robot variant (IRB 6600) and the desired track model to the station. See *Adding the track to the system on page 68*.

NOTE!

1. In the **Select Library** group, click **Other** to import a different track motion library.
2. Click **OK**. When asked if you want to restart the system, answer **Yes**. Close the **System Configuration** window.

2.5 The VC

2.5.1. Starting a VC

Overview

RobotStudio uses virtual controllers for running the robots. Virtual controllers can run both systems for real robots and specific virtual systems for testing and evaluation purposes. A virtual controller uses the same software as the controller to execute the RAPID program, to calculate robot motions and to handle I/O signals.

When starting a virtual controller, point out which system to run on it. Since the system contains information about the robots to use and important data such as robot programs and configurations, it is important to select the right system for the station.

Starting a VC

The table below describes the different ways a virtual controller may start:

Startup	Description
Automatic, when creating a station	In most cases, a VC is automatically started when you create a new station. Library files for the robots used by the system are then imported to the station.
Automatic, when adding a system to an existing station	If your station uses several systems or if you started with an empty station, you can add systems to an open station. Library files for the robots used by the the systems are then imported to the station.
Manually, when connecting to an imported library	If you have manually imported a robot library you want to use with a system, instead of importing a new library at startup, you can connect this library to a controller. If you have manually imported a robot library you want to use with a system, instead of importing a new library at startup, you can connect this library to a controller. A library may only be connected to a single-robot system and may not be already connected to another VC.

Continues on next page

Adding a system

To start a system in a new virtual controller and automatically import the robots used by the system, follow these steps:

1. • If you have the **Select system** dialog box open, continue with the next step.
   • If you do not have the **Select system** dialog box open, go to the **Controller** menu and click **Add System**.
2. Depending on whether the system to start on the virtual controller is ready or not, do one of the following:

Activity	Description
Create and start a copy of an existing system This is recommended when you plan to add station-specific configurations or programs.	From the **Select system pool** list, select the folder on your PC that contains the system to use. Select the system to copy from the **Systems Found** list and click **Copy**. In the **Copy System** dialog box, enter a name for the system and click **OK**.
Start an existing system as is This is useful when you already have created a specific system for the station. It is also useful if you do not plan to create station-specific configurations or programs and want a quick start.	From the **Select system pool** list, select the folder on your PC that contains the system to use.
Modify and start an existing system This is the way to add or change options in the systems, for example, adding or replacing robots and external axes.	From the **Select system pool** list, select the folder on your PC that contains the system to modify. Select the system to modify from the **Systems Found** list and click **Modify**. This starts the **System Builder** from which you make the modifications. For detailed information, see *Modifying a system on page 145*.
Build and start a new system This option is used for building and starting a completely new system, either from RobotWare keys or from virtual keys.	Click **Create** to start the **System Builder** from which you build the new system. For detailed information about creating the system, see *Building a new system on page 142*.

3. From the **Systems Found** list, select the system to use and click **Start**.

2.5.2. Restarting a VC

Overview

This section describes when and how to restart a VC in RobotStudio. See also *Restart on page 287*.

Prerequisites

The following are the prerequisites for restarting a controller:

- You must have Write access to the controller you are restarting.
- For the advanced restart methods X-start and C-start, you must have access to the controller's FlexPendant.

When to restart a virtual controller

Some operations require a restart of the controller to take effect. When working in RobotStudio, you will be notified when a restart is necessary.

Warm restart

Typically, you need to make an ordinary warm restart of a virtual controller when:

- you have changed the baseframe of any of the robots belonging to that virtual controller.
- you have changed the robot's configuration, either with the Configuration Editor or by loading new configuration files.
- you have added new options or hardware to the system.
- A system failure has occurred.

Cold restart

A cold restart is useful for restoring the system running on a virtual controller to the state it was created in. The cold restart deletes all RAPID programs, data and custom configurations that have been added to the system.

Advanced restarts

In addition to warm and cold restarts, several advanced restarts are available. See *Advanced restart options on page 66*.

Advanced restart options

The controller can be restarted with the following advanced restart options:

Option	Description
I-Start	Restarts the controller with the current system and the default settings. This method discards the changes made to the robot's configuration. It reverts the current system to the state it had when it was installed on the controller (an empty system).

Continues on next page

Option	Description
P-Start	Restart the controller with the current system and reinstall RAPID. This method deletes all RAPID program modules. It can be useful if the system has changed in such a way that the programs no longer are valid, for instance if system parameters used by the program are changed.
X-Start	Saves the current system and starts the boot application. This method saves the current system, with the current settings, and starts the boot application from which you can choose a new system to start with. You can also configure the controller's network settings from the boot application.
C-Start	Deletes current system and start boot application. This method deletes the current system and starts the boot application from which you can choose a new system to start with. You can also configure the controller's network settings from the boot application.
B-Start	Restarts the controller with the current system and the last known good settings.This method restores changes made to the robot's configuration to a previously good state.

2.6 Station components

2.6.1. Importing a station component

Importing a robot model

This is how to import a robot model without a controller to your station.

A robot which is not connected to a controller cannot be programed. To import a robot connected to a virtual controller, configure a system for the robot and start it in a virtual controller, see *Building a new system on page 142* and *Starting a VC on page 64*, respectively.

To import a robot model, in the **Home** tab, click **Robot System** and then select a robot model from the gallery.

Importing a tool

A tool is a special object, for example, an arc weld gun or a gripper, that operates on the work piece. For achieving correct motions in robot programs, the parameters of the tool have to be specified in the tool data. The most essential part of the tool data is the TCP, which is the position of the tool center point relative to the wrist of the robot (which is the same as the default tool, *tool0*).

When imported, the tool will not be related to the robot. So in order for the tool to move with the robot, you must attach it to the robot.

To import a tool, in the **Home** tab, click **Tool** and then select a tool from the gallery.

Importing a positioner

To import a tool, in the **Home** tab, click **Positioner** and then select a positioner from the gallery.

Adding the track to the system

To select the model of the external axis to use, follow these steps:

1. Start the system in a virtual controller, either in a new empty station or in an existing station, see *Robot System on page 188*.
2. In the **Layout** browser, select the system to add the track to.
3. On the **Offline** tab, click **System Configuration**.

Continues on next page

4. Click **Add** to add parameters for the track to the system. Browse to the parameter file (.cfg) for the track to add and click **Open**.

   If you have a specific parameter file for you track, use that one. Otherwise, parameter files for some standard tracks are delivered with the RobotStudio installation. These can be found in the folder *ABB Library/ Tracks* in RobotStudio's installation folder. The *ABB Library* folder can also be opened from the Quick access pane at the left of the Open dialog box used for adding parameter files.

   The file name of each parameter file tells which tracks it supports. The first part tells the length of the track and the second the number of tasks.

   For example, the file TRACK_1_7.cfg supports all tracks with the length 1.7 meters in systems with one single task. For Multimove systems or other systems with several tasks, use the configuration file with the matching number of tasks.

   For example, if the track length is 19.9 m and the robot attached to the track is connected to task 4 of the MultiMove system, then select TRACK_19_9_Task4.cfg file.
5. In the **System Configuration** window, click **OK**. When asked if you want to restart the system, answer **Yes**.
6. During the restart a list of all tracks compatible with the configuration file is displayed. Select the one to use and click **OK**.

   After the restart the track appears in the station. Continue with attaching the robot to the track.

Importing a library, geometry or piece of equipment

A library component is a RobotStudio object that has been saved separately. Normally, components in a library are locked for editing.

A geometry is CAD data which you can import to use in RobotStudio. For a list of importable CAD formats, see *Libraries, geometries and CAD files on page 26*.

To import a library, geometry or piece of equipment, see *Import Library on page 187*.

2.6.2. Converting CAD formats

Overview

A CAD converter is installed together with RobotStudio by default. In most cases you do not have to convert CAD files before importing them to RobotStudio, but the CAD converter might be useful for converting several files at once, or for converting with custom settings.

Prerequisites

Most of the file formats require separate licenses, see *Libraries, geometries and CAD files on page 26* for more information.

Starting the CAD converter

Click **Start** menu, point to **Programs**, **ABB Industrial IT**, **Robotics IT** and click **CAD Converter**.

Converting CAD files

To convert CAD files, follow these steps:

1. Click **Add files** and select the files to convert. Optionally, click **Add files**, again to add more files from another location.

   Each file is now added to a row in the grid.
2. Optionally, change the suggested file name or target format by clicking that column for the file to change.
3. In the **Target directory** box, specify the folder in which to save the new files.
4. Optionally, click **Settings** and change the settings for the conversion. For details about the conversion settings, see *Conversion settings on page 70*.
5. Click **Convert Files**.

Conversion settings

The table below describes the settings for the conversion:

Setting	Description
Acis save file format	Select which version of ACIS to save to when using ACIS as target format.
Enable Healing	Controls whether the conversion engine attempts to heal geometric entities. Only supported for specified formats.
Translate hidden/no-show entities	Controls whether the hidden entities are translated or discarded. Only supported for specified formats.
VRML/STL Scale factors	VRML and STL are often created in units that RobotStudio does not expect; they thus need to be resized.
Delete all generated log files on exit	Makes the CAD converter delete log files when exiting.

2.6.3. Troubleshooting and optimizing geometries

Overview

The characteristics of the geometries and CAD models in the station may have great effect on your work in RobotStudio, both in aspects of making the objects easier to program as well as enhancing simulation performance.

Below are some guidelines for troubleshooting geometries.

Trouble	Information
The pointer snaps to the wrong parts of the objects when selecting in the graphics window	This problem might be caused by wrong snap mode settings, imprecise selecting, hidden or lack of geometrical information. To resolve these problems, do the following: • Check the selection level and snap mode settings. For more information, see *Selecting an item on page 47*. • When making the selection, zoom and rotate the object so that you are sure to click *inside* the object. • Check if the object has hidden details that might affect the snapping. Remove details that are not necessary for your programming or simulation. For more information, see *Modifying a part on page 74*. • Some file formats only contain a graphical representation and no geometrical data. Import the geometry from a file format that also contains geometrical data. For more information, see *Libraries, geometries and CAD files on page 26*.
The graphics window redraws or updates slowly	This might be due to the performance of your computer not being high enough for the size of the geometry files in your station. To reduce the size of the geometry files, do any of the following: • Use a lower detail level for rendering the geometry. For more information, see *Graphic Appearance on page 333*. • Blends, chamfers and holes can be automatically simplified by using the Defeature function. This can greatly reduce graphic complexity, speed up simulations and reduce memory usage. For more information, see *To defeature an object*. • Check if the object has unnecessary details. Remove details that are not necessary for your programming or simulation. For more information, see *Modifying a part on page 74*.

Continues on next page

Trouble	Information
Parts of the geometry are not visible	If parts of the geometry are not visible from some views, a probable cause is that the object is made up of 2D surfaces and the option *Backface culling* is on. Backface culling means that the faces of the object are only visible from the front, and if the object (or any of its faces) is oriented differently, they will not be visible. To correct the problem, do one of the following: • Switch to modeling mode and invert the direction of the face that is not displayed correctly. This not only corrects the display, it also decreases the chance of faulty orientations during graphical programming. For more information, see *Invert on page 336* or *To invert the direction of all faces of a part on page 72*. • Turn backface culling off for the specific object. This makes the object display correctly, but does not affect the direction of the face, which might cause problems if the face will be used for graphical programming. For more information, see *To deactivate backface culling for a single object on page 72*. • Turn backface culling off for all objects in the station. This makes the objects display correctly, but does not affect the direction of the face, which might cause problems if the face will be used for graphical programming. It also decreases the performance of the graphic handling. For more information, see *To change the generic setting for backface culling on page 72*.

To invert the direction of all faces of a part

To invert the direction of all faces of a part, follow these steps:

1. Select the part on which faces you want to invert the directions.
2. On the **Modify** menu, click **Graphic Appearance**.
3. On the **Rendering** tab, click **Flip normals** and then click **OK**.

To deactivate backface culling for a single object

To change the backface culling setting for a single object, follow these steps:

1. Select the part for which you want to change the backface culling setting.
2. On the **Modify** menu, click **Graphic Appearance**.
3. On the **Rendering** tab, clear the **Backface culling** check box and then click **OK**. The faces of the object will now be displayed even if the generic setting for backface culling is on.

To change the generic setting for backface culling

The generic setting for backface culling affects all new objects and existing objects that do not have backface culling specifically deactivated.

1. On the **Application** menu, click **RobotStudio Options**.
2. On the **Navigation pane** to the left, select **Graphics: Performance**.
3. On the **Performance** page, select or clear the **Cull back-facing triangles** check box and then click **OK**.

2.7 Modeling

2.7.1. Objects

Overview

This section describes how to create or modify geometrical objects.

Creating a frame

A frame is a generic coordinate system that you can use as reference when positioning objects. Generic frames can also be converted to special kinds of coordinate systesm, like workobjects or tool center points.

For procedures, see *Frame on page 192* and *Frame from Three Points on page 193*.

Creating a solid

With the create solids commands you can create and build models of objects you do not have CAD files or libraries for. With the create solids commands you create primitve solid bodies; these can later be combined to more complex bodies.

For procedures, see *Solid on page 228*.

Creating a surface

For procedures, see *Surface on page 232*.

Creating a curve

When creating paths with targets based on the object geometries, curves are the geometrical objects that RobotStudio uses. For example, if you want the robot to run along the edge of an object, you can first create a curve along the border and then generate a complete path along that curve, instead of manually finding and creating the necessary targets.

If the CAD model/geometry of the work piece does not already contain curves, you can create the curves i RobotStudio.

For procedures, see *Curve on page 234*.

Modifying a curve

When creating paths with targets based on the objects geometries, curves are the geometrical objects that RobotStudio uses. By optimizing the curves before starting programming, you reduce the touch-up of the generated paths.

For procedures, see *Modify Curve on page 344*.

Creating a border

For procedures, see *Border on page 239*.

Creating a line from normal

A line can be created as a new part and body perpendicular to a surface.

For a procedure, see *Line from Normal on page 245*.

Continues on next page

Extruding a surface or curve

Curves and surfaces and curves can also be extruded to 3D objects, which may then be converted to solids. You can extrude along either a vector or a curve.

For procedures, see *Extrude Surface or Curve on page 244*.

Modifying a part

When you import a geometry or create an object, it will be one part. A part can, however, contain several bodies. In RobotStudio's modeling mode you can modify the parts by adding, moving and deleting the bodies.

To modify a part, follow this step:

1. In the **Modeling** browser, expand the node for the part to modify. Then modify the part by doing any of the following:

To	Do this
Delete a body	Select the body and press the DEL key.
Move a body from one part to another	Drag the body or use the **Copy** and **Paste** commands on the **Edit** menu.
Move one body relative to the others	Select the body and then move it using any of the ordinary commands for moving objects, see *Placing objects*.

Modifying a library component

As external files, libraries are merely linked from a station. Therefore, to modify an imported library component, the link must first be broken and later reestablished. For procedures, see *The Library Group on page 339*.

2.7.2. Mechanisms

Workflow

This information topic describes how to create a new mechanism, that is, a graphical representation of a robot, tool, external axis or device. The various parts of a mechanism move along or around axes.

Creating a mechanism is dependent upon skillful construction of the main nodes of the tree structure. Four of these—links, joints, frames/tools and calibration—are initially marked red. As each node is configured with enough subnodes to make it valid, the marking turns to green. As soon as all nodes have become valid, the mechanism will be considered compilable and can be created. For additional validity criteria, see the table below.

Node	Validity criteria
Links	• It contains more than one subnode. • The BaseLink is set. • All link parts are still in the station.
Joints	• At least one joint must be active and valid.
Frame/tool Data	• At least one frame/tool data exists. • For a device, no frames are needed.
Calibration	• For a robot, exactly one calibration is required. • For an external axis, one calibration is required for each joint. • For a tool or device, calibrations are accepted, but not required.
Dependencies	• None.

The modify mode of the Mechanism Modeler has two purposes: to enable modification of an editable mechanism in its tree structure, and to complete the modeling of a new or modified mechanism.

It is recommended to configure each main node in the tree structure from the top down. Depending on its current status, right-click or double-click a node or subnode to add, edit or remove it.

For procedures, see *Create Mechanism on page 247*.

2.7.3. Tools and tooldata

Overview

To simulate the robot tool, you need tooldata for the tool. If you import a predefined tool or if you create a tool using the **Create Tool Wizard**, the tooldata is automatically created; otherwise, you have to create the tooldata yourself.

The tooldata simplifies the programming work with respect to the different tools that may come in use. Defining separate sets of tooldata for different tools makes it possible to run the same robot program with different tools: only the new tooldata has to be defined. The tooldata contains the information required for moving and simulating the tool.

Two methods for manipulating tooldata in RobotStudio are as follows:

- Create or modify tooldata, see *Tooldata on page 196* and *Modify Tooldata on page 352*, respectively. This will create all data necessary for programming, but there will be no visual tool during the simulation.
- Create tooldata for an existing geometry, *Create Tool on page 253*.

Creating and setting up a stationary tool

This information topic describes how to create a stationary tool. For information about creating a robot hold tool, see *Create Tool on page 253*.

Using a stationary tool, the robot holds and moves the work piece in relation to the tool. Thus, both the tooldata and the workobject must be set up correctly.

To create the tooldata for a stationary tool, follow these steps:

1. Import the geometry or library that represents the tool, see *Import Geometry on page 191*.

   If you do not have the geometry or library at hand but know the position, you can skip this step. The tool will be programable, but not visible in the station.
2. Create the tooldata for the tool, see *Tooldata on page 196*.

   Make sure to set the **Robot holds tool** option to **false**.
3. Create a workobject that is moved by the robot. see *Workobject on page 195*.

   Make sure to set the **Robot holds workobject** option to **true**.
4. If you have a geometry or library component for the work piece, attach it to the robot, see *Attach to on page 323*.

2.7.4. Setting the local origin of an object

Overview

Each object has a coordinate system of its own called local coordinate system in which the object dimensions are defined. When the object's position is referred from other coordinate system, it is the origin of this coordinate system that is used.

With the Set Local Origin command you reposition the object's local coordinate system, not the object itself.

For a procedure, see *Set Local Origin on page 365*.

2.8 Placement

2.8.1. Placing objects

Overview

To achieve the required layout of your station, you need to import or create objects, place them accordingly and, if applicable, attach them to other objects.

Placing objects means setting their position and rotation. If the objects are to be attached to robots or other mechanisms, they will be placed at their attachment point automatically.

The following table describes the actions relating to placement:

Actions	Description
Placing an object	To place an object is to put the object in the required position in the station, see *Place on page 356* and *Set Position on page 367*.
Rotating an object	The objects in the station can be rotated to achieve the required layout, see *Rotate on page 361*.
Measuring distance or amgles	The measurement functions calculates distances, angles and diameters between points you select from the graphics window. When using measurements, results and instructions on how to proceed are displayed in the **Output** window, see *The Measure Group on page 246*.
Creating a component group	A component group groups related object in the browser, see *Component Group on page 226*.
Attaching or detaching an object	Objects that are to be used by the robots in any way, such as tools, need to be attached to the robot, see *Attach to on page 323* and *Detach on page 331*.
Jogging a robot	Robots can be placed by jogging. The robot axes can also be postioned by jogging, see *Jogging mechanisms on page 84*.
Modifying the task frame	Modifying the task frame repositions a controller and all its robots and equipment in the station. By default the controller world and the station world coordinate system coincide. This is convenient when building a station with one single controller. For a procedure, see *Set Task Frames on page 293*. However, when you have several controllers in one station, or need to reposition a controller in an existing station, you need to modify the *System Configuration on page 294*.
Modifying the baseframe position	Modifying the baseframe position sets an offset between the controller's world coordinate system and the baseframe of the mechanical unit.This is necessary when having several mechanical units belonging to one controller, for example, several robots in MultiMove systems or when using positioner external axes. For a procedure, see *System Configuration on page 294*.

2.8.2. Placing external axes

Overview

When starting a system with a track or positioner external axis in a RobotStudio station, you have to set up the system to load a model for the track or positioner and get the motions to work properly.

Prerequisites

The system shall be created with support for track or positioner external axes, see *A system with support for one robot and one positioner external axis on page 155*.

Attaching the robot to the track

To attach the robot to the track, follow these steps:

1. In the **Layout** browser, drag the robot icon and drop it on the track icon.
2. On the question *Should the robot be coordinated with the track?*, answer **Yes** to be able to coordinate the track's position with that of the robot in robot programs. To program the track and the robot independently, answer **No**.
3. When asked if you want to restart the system, answer **Yes**.

   The track is now added to the system and ready to be programmed, see *Programming external axes on page 109* for more information on how to program the track.

CAUTION!

If the system is cold started , the setup is deleted and the procedures described here must be performed again.

Placing the positioner in the station

To place the positioner in the station, follow these steps:

1. Move the positioner to the desired position using any of the ordinary functions for placing and moving objects, see *Placing objects*.
2. Modify the baseframe position of each mechanical unit of the positioner except the INTERCH unit, if it exists. When asked if you want to restart the system, answer **Yes.**

   See *Baseframe position* for detailed instruction.

   After the restart the system is updated with the positioner's new location. Continue attaching fixtures and workobjects to the positioner.

Attaching objects to the positioner

To program robot motions on an object that is held by the positioner, the targets must be created in a workobject that is attached to the positioner. For a complete visual simulation, CAD models that are moved by the positioner should also be attached. To attach the objects, follow these steps:

1. Import the models of the fixture and the work piece if you do not have them in the station already, see *Importing a station component on page 68*.
2. Attach the fixture to the positioner, see *Attaching and detaching objects on page 48*. When asked whether to keep the current position, answer **No**.

   If the positioner has several stations, you will be asked which one to attach the object to.

Continues on next page

3. Attach the work piece to the fixture. When asked whether to keep the current position, answer **No**.
4. Attach the workobject in which you will program the work piece to either the fixture, the work piece or the positioner. If you have defined calibration positions on either the work piece or the fixture, it is a good practice to use that object. When asked whether to keep the current position, answer **No**.

   The positioner is now set up and ready to be programed, see *Programming external axes on page 109* for more information.

TIP!

If the positioner is of an interchangeable type with several stations, you can either attach individual fixtures, work pieces and workobjects to each station flange, or you can use one set of objects that you attach and detach to the different flanges by events.

CAUTION!

If the system is cold started, the setup is deleted and the procedures described here must be performed again.

3 How to program robots

3.1. Workflow for programming a robot

Overview

In most cases, going through the workflow from start to finish is recommended, even if it possible to work in other sequences as well.

Synchronizing will save and load text files containing RAPID modules, and create RAPID programs from your station.

Prerequisites

Before creating a program for your robot, you should set up the station, including robots, work pieces and fixtures, in which your robot will work.

Programming a robot

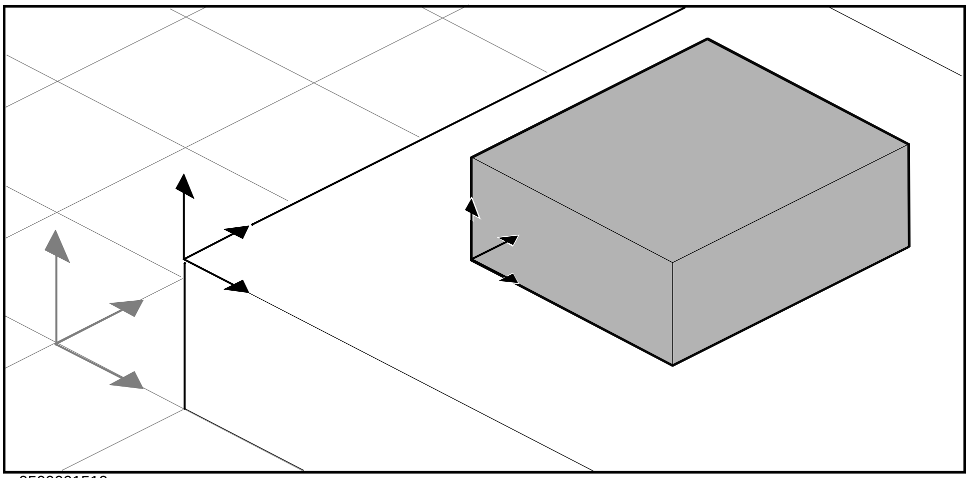
The table below describes the workflow for programming a robot to perform the task you require.

Task	Description
Create targets and paths	Create the targets and paths the robot requires to perform the work tasks. To create targets and paths, do one of the following: • Create a curve to match your required shape. Then use the **Create path from curve** command to generate a path, complete with targets, along the shape you have created. See *Curve on page 234* and *Path from Curve on page 202*. • Create targets at the requested positions, then create a path and insert the created targets into it. See *Create Target on page 198*, *Teach Target on page 197* and *Empty Path on page 201*.
Check the target orientations	Make sure that the targets are oriented in the most efficient way for the tasks to be performed. If not, reorient the targets until you are satisfied. See *Orientations on page 91*.
Check reachability	Check that the robot and tool reach all targets in the path. See *Testing positions and motions on page 100*.
Synchronize the program to the virtual controller	Generates RAPID code from the RobotStudio items and enables the program to be simulated.
Perform text-based editing	If you need to edit the instructions or data created by RobotStudio, you can start the Program Editor. See *Using the RAPID editor on page 113*.
Collision detection	Check that the robot or tool does not collide with the surrounding equipment or the fixtures. If it does, adjust the placements or orientations until no collisions occur. See *Detecting collisions on page 119*.
Test the program	Test the program by moving along the paths. See *Testing positions and motions on page 100*.

3.2. Workobjects

Creating a workobject

A workobject is a coordinate system used to describe the position of a work piece. The workobject consists of two frames: a user frame and an object frame. All programed positions will be related to the object frame, which is related to the user frame, which is related to the world coordinate system.



xx0500001519

For a procedure, see *Workobject on page 195*.

Modifying a workobject

For a procedure, see *Modify Workobject on page 353*.

Converting a frame to a workobject

You can create a new workobject from an existing frame. The converted workobject gets the same name and position as the selected frame.

For a procedure, see *Convert Frame to Workobject on page 328*.

Creating a frame by points

You can create a frame by specfiying points on the axes of the coordinate system and letting RobotStudio calulate the placement and orientation of the frame's origin.

For a procedure, see *Frame from Three Points on page 193*.

3.3. Jogging mechanisms

Jogging a robot

To check if the robot can reach all positions on the work piece, you can jog the TCP or the joints of the robot, either with the freehand commands of through dialog boxes. Jogging the robot close to its boudaries is best done with the latter method.

To	Procedure
Jog the joints of a robot	For freehand, see *Jog Joint on page 221*. For a dialog box, see *Mechanism Joint Jog on page 340*.
Jog the TCP of a robot	For freehand, see *Jog Linear on page 222*. For a dialog box, see *Mechanism Linear Jog on page 342*.

Prerequisites

To jog the TCP of a robot, the robot's VC must be running.

Jogging several mechanisms

Function	Description
Mutlirobot jog	When using multirobot jog, all selected mechanisms will follow the TCP of the one being jogged. Multirobot jog is available for all kinds of jogging. See *MultiRobot Jog on page 223*.
Jogging with locked TCP	When jogging a mechanism that moves a robot (like a track external axis) with locked TCP, the robot will reposition so that the position of the TCP does not change, even though its baseframe is moved. When jogging an external axis that moves the work object with locked TCP, the robot will reposition so that its TCP follows the work object in the same way as when using multirobot jog. Locked TCP is available when jogging a mechanism that belongs to the same task as a robot. See *Mechanism Joint Jog on page 340*.

3.4. Targets

Creating a target

You can create a new target manually either by entering the position for the target in the **Create Target** dialog box or by clicking in the graphics window.

The target will be created in the active workobject.

For a procedure, see *Create Target on page 198*.

Creating a jointtarget

A jointtarget is a specification of the position for the robot axes.

For a procedure, see *Create Jointtarget on page 200*.

Teaching targets

You can create a new target by jogging the robot and teaching a target at the active TCP. Taught targets will be created with the axis configuration used when jogged to the target.

The target will be created in the active workobject.

For a procedure, see *Teach Target on page 197*.

Modifying a target position

By using the modify position command you can modify the position and rotation of a target.

For procedures, see *Set Position on page 367* and *Rotate on page 361*, respectively.

Modifying a target with ModPos

The position of an existing target can be modified by jogging the robot to a new, preferred position. By selecting a move instruction for the target in a path, the ModPos command can be used to move the target to the TCP of the active tool.

When ModPos is executed, the target, referenced to by the move instruction, will be updated with the following information:

- position and orientation corresponding to the TCP of the active tool
- the current configuration of the active robot
- the current position and orientation values of all active external axes for the active robot

NOTE: To jog a robot linearly, a virtual controller must be running for that robot. For detailed information, see *Starting a VC on page 64*.

Continues on next page

Renaming targets

With this command you can change the name of several targets at once. You can either rename targets individually, or you can rename all targets in one or several paths at once.

The new target names will consist of an optional prefix, an incremental number and an optional suffix.

For a precedure, see *Rename Targets on page 359*.

When renaming targets, make sure that the new targets conform to the naming rules. The target names must:

- start with an alphabetical character in the ISO 8859-1 encoding (that is, an ordinary letter from the English alphabet)
- be shorter than 16 characters
- not be empty strings
- not contain any characters illegal in RAPID. See the *RAPID reference manual* for details.

Removing unused targets

If deleting or changing paths or move instructions during programming, you might end up with large numbers of targets that are no longer used in any instructions. To make the workobjects and their targets easier to grasp, you can delete all unused targets.

For a procedure, see *Remove Unused Targets on page 358*.

3.5. Paths

Creating an empty path

A path is a sequence of targets with move instructions that the robot follows. An empty path will be created in the active task.

For a procedure, see *Empty Path on page 201*.

Creating a path from curve

If the work piece has curves or contours that correspond to the path to be created, you can create the paths automatically. The create path from curve command generates paths, complete with targets and instructions along existing curves.

The path will be created in the active task.

The orientation of the targets that will be created will be according to the settings of the approach/travel vectors in the **Options** dialog box.

To create a path from a curve, the curve must have first been created in the station.

For a procedure, see *Path from Curve on page 202*.

Setting robot axis configuration for paths

The robot axis configuration specifies the position of the axes as the robot moves from target to target, when multiple solutions are possible. This is necessary for executing move instructions using configuration monitoring.

Taught targets have validated configurations, but targets created in any other way do not. Also, targets that are repositioned lose their configuration. In RobotStudio, targets without a valid configuration are marked with a yellow warning symbol. See *Robot axis configurations on page 24* for more information about configurations.

To set a configuration for all targets in a path, see *Auto Configuration on page 324*.

To set a configuration for a single target, see *Configurations on page 326*.

Reversing paths

The reverse path commands change the sequence of targets in the path so that the robot moves from the last target to the first. When reversing paths, you can reverse either the target sequence alone or the entire motion process.

For procedures, see *Reverse Path on page 360*.

Continues on next page

NOTE!

When reversing paths, the original paths are deleted. If you want to keep them, make copies before reversal.

NOTE!

When reversing paths, only move instructions are handled. Action instructions, if any exist, have to be inserted manually after the reversal.

Rotating paths

With the rotate path command you can rotate complete paths and move the targets used by the paths accordingly. When rotating paths, the included targets will lose their axis configurations, if any have been assigned.

A frame or target must exist at the position to rotate around before starting the rotate path command.

For a procedure, see *Rotate Path on page 362*.

Translating a path

The translate path function moves a path and all included targets.

For a procedure, see *Translate Path on page 369*.

Compensating paths for tool radius

You can offset a path so that it compensates for the radius of a rotating tool. Since the targets in the path are moved, they will lose their axis configurations, if any have been assigned.

For a procedure, see *Tool Compensation on page 368*.

Continues on next page

Interpolating a path

The interpolate functions reorient the targets in a path so that the difference in orientation between the start and end targets is distributed evenly among the targets in between. The interpolation can be either linear or absolute.

Linear interpolation distributes the difference in orientation evenly, based on the targets' positions along the length of the path.

Absolute interpolation distributes the difference in orientation evenly, based on the targets' sequence in the path.

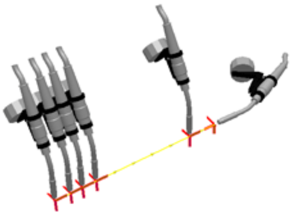
Below are examples of the difference bewteen linear and absolute interpolation.

The interpolate functions reorient the targets in a path so that the difference in orientation between the start and end targets is distributed evenly among the targets in between. The interpolation can be either linear or absolute.

For a procedure, see *Interpolate Path on page 335*.

No interpolation

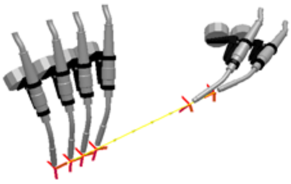
This is the path before any interpolation. Note that the last target is oriented differently than the others.



xx0500002029

Linear interpolation

This is the same path after linear interpolation.



xx0500002030

Note that the targets are oriented based on their placement relative to the start and end targets.

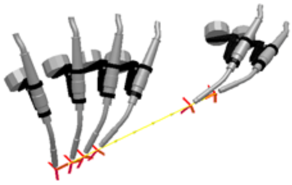
If a target were moved and you reran the linear interpolation, it would be reoriented according to its new position.

If new targets were inserted between the existing ones and you reran the linear interpolation, it would not affect the orientation of the existing targets.

Continues on next page

Absolute interpolation

This is the same path after absolute interpolation



xx0500002031

Note that the targets are orientated based on their sequence in the path: each target has been reoriented equally, regardless of its place.

If a target were moved and you reran the absolute interpolation, it would not affect the orientation.

If new targets were inserted between the existing ones and you reran the absolute interpolation, it would change the orientation of all targets.

Mirroring a path

The mirror path function mirrors all motions instructions and their targets to a new path.

For a procedure, see *Mirror Path on page 343*.

3.6. Orientations

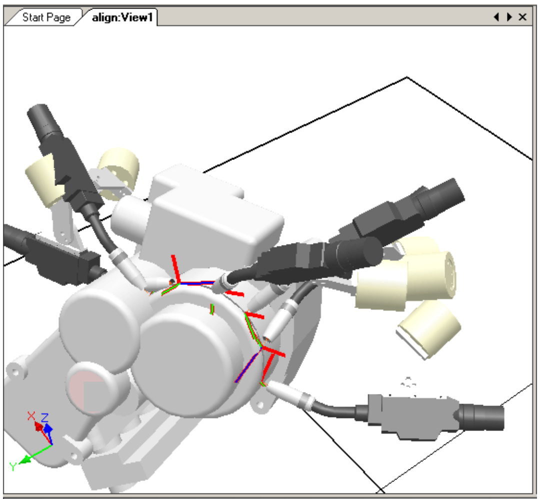
Overview

This is an overview of the tools for automating the modification of target orientations.

When creating paths from curves in RobotStudio, the orientation of the targets depends on the characteristics of the curves and the surrounding surfaces. Below is an example of a path with unordered target orientations and examples of how the different tools have affected the targets.

Unordered orientations

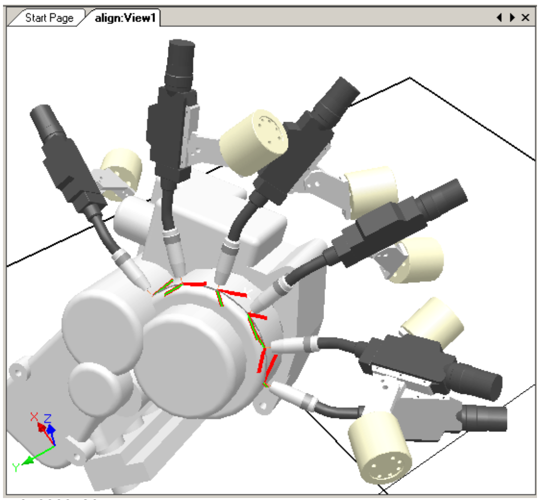
In the path below, the target orientations are unordered. The function View tool at target has been used for illustrating how the targets point in different directions.



xx0500001864

Effect of target normal to surface

In the picture below, the targets, which previously were orientated randomly, have been set normal to the flat round surface at the right side of the path. Note how the targets' Z axis has been orientated normal to the surface; the targets have not been rotated in the other directions.



xx0500001865

Continues on next page

Continued

Setting a target normal to surface

To set a target orientation normal to a surface is to make it perpendicular to the surface. The target can be oriented normal to the surface in two different ways:

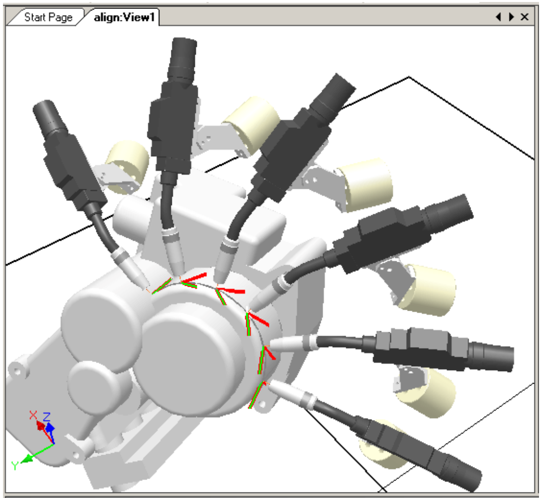
- The entire surface can be used as a reference for the normal. The target will be oriented as the normal to the closest point at the surface. The entire surface is the default surface reference.
- A specific point on the surface can be used as the reference for the normal. The target will be orientated as the normal to this point, regardless of whether the normal to the closest point at the surface has another orientation.

Objects imported without geometry (for example, .jt files) can only refer to specific points on the surface.

For a procedure, see *Set Normal to Surface on page 366*.

Effect of align target orientation

In the picture below, the targets, which were previously orientated with the Z axis normal to the surface but with the X and Y axes orientated randomly, have been organized by aligning the targets' orientation around the X axis with the Z axis locked. One of the targets in the path has been used as reference.



xx0500001866

Aligning a target orientation

With the align target orientation command you align the rotation of selected targets around one axis without changing the rotation around the others.

For a procedure, see *Align Target Orientation on page 322*.

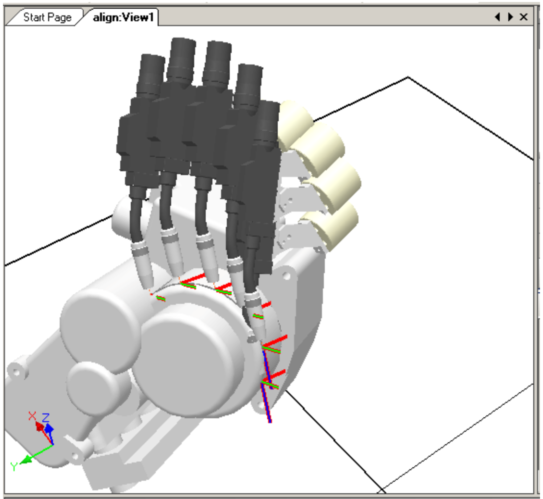
Continues on next page

TIP!

You can also align ordinary frames in the same way.

Effects of copy and apply orientation

In the picture below, the targets, which were previously oriented randomly, have been organized by copying the exact orientation of one target to all the others. This is a quick way to fix workable orientations for processes where variations in approach, travel, or spin directions either do not matter or are not affected, due to the shape of the work piece.



xx0500001867

Copying and applying an orientation for objects

To transfer an orientation from one object to another is an easy way to align different frames for simplifying the programming of the robot. Target orientations may also be copied.

For procedures, see *Copy / Apply Orientation on page 330*.

3.7. RAPID Instructions

Move and action instructions

For RAPID programming, RobotStudio's main advantage is in the area of motion programming.

A move instruction is an instruction for the robot to move to a specified target in a specified manner. With RobotStudio, you can create move instructions in three ways:

Method	Description
Create a move instruction based on an existing target	Creates move instructions based on one or several targets selected in the **Paths&Targets** browser. For a procedure, see *Add to Path on page 320*.
Create a move instruction and a corresponding target	Creates a move instruction and a corresponding target at once. The position of the target can either be selected from the graphics window or typed numerically. For a procedure, see *Move Instruction on page 214*.
Teach a move instruction	Teaching a move instruction creates a move instruction and a corresponding target at the robot's current position. Teaching a move instruction also stores the current configuration with the target. For a procedure, see *Teach Instruction on page 213*.

In addition to move instructions, you can also create and insert action instructions from RobotStudio. An action instruction is an instruction other than a move instruction that can, for example, set parameters, or activate or deactivate equipment and functions. The action instructions available in RobotStudio are limited to those commonly used for affecting the robot's motions. For inserting other action instructions or another kind of RAPID code in the program, use the Program Editor. For a procedure, see *Action Instruction on page 215*.

The table below lists the action instructions that can be created. For details, see the RAPID Reference Manual.

Action instruction	Description
ConfL On/Off	ConfL specifies whether to monitor the robot's configurations during linear movements. When ConfL is set to Off, the robot may use another configuration than the programed one for reaching the target during program execution.
ConfJ On/Off	ConfJ specifies whether to monitor the robot's configurations during joint movements. When ConfJ is set to Off, the robot may use another configuration than the programed one for reaching the target during program execution.
Actunit *UnitName*	Actunit activates the mechanical unit specified by *UnitName*.
DeactUnit *UnitName*	Deactunit deactivates the mechanical unit specified by *UnitName*.
ConfJ On/Off	ConfJ specifies whether to monitor the robot's configurations during joint movements. When ConfJ is set to Off, the robot may use another configuration than the programed one for reaching the target during program execution.
Actunit *UnitName*	Actunit activates the mechanical unit specified by *UnitName*.
DeactUnit *UnitName*	Deactunit deactivates the mechanical unit specified by *UnitName*.

Continues on next page

Modifying an instruction

Most instructions have arguments that specify how the instruction shall be carried out. For example, the MoveL instruction has arguments that specify the speed and accuracy with which the robot moves to the target.

For a procedure, see *Modify Instruction on page 350*.

NOTE!

Some arguments are read from the virtual controller. If the virtual controller has not been started, only the arguments stored in the station can be modified.

Converting to move circular

To create a circular motion to an instruction target, you must convert the motion type to circular motion (that is, MoveC in RAPID).

A circular motion is defined by two motion instructions, where the first is the via-point and the second contains the end point of the circular motion.

The circular motion can only be used for open circular arcs, not for closed circles. To create a path for a closed circle, use two circular motions.

For a procedure, see *Convert to Move Circular on page 329*.

Creating RAPID instructions for setting I/O signals

For controlling I/O signals in the robot program, you use RAPID commands that set the signals. These require that you first create instruction templates for the instructions that set the signals. See the *RAPID reference manual* for details about the instructions that control I/O signals.

To add RAPID instructions that set I/O signals, follow these steps:

1. Synchronize the system in which you want to add the instructions to the virtual controller, see *Synchronization on page 112*.
2. In programming mode, select the module for editing, right-click it and then click **Edit program**.
3. In the program editor, add the instructions for setting the signals.
4. When you are done adding instructions, synchronize the task and paths from the Virtual Controller back to the station.

Continues on next page

Using cross-connections and groups for setting I/O signals

You can also create cross-connections and signal groups, which make one signal set the value of several other signals. See the *System parameters reference manual* for details about cross-connections and groups.

To make one signal set several others, follow these steps:

1. Request write access, and then open the configuration topic I/O in the configuration editor. Add configure instances for the cross-connections and groups to create.

Instruction templates

Instruction templates contain predefined sets of argument values that are applied to the instructions you create using the template. You can create templates for all instructions in the system running on the virtual controller. To see which instructions are available and what their arguments do, see the RAPID reference manual for your RobotWare version and the reference sections in manuals for software options, if you have any installed on the system.

Move instruction templates are always part of *process templates*. The process templates contain one instruction template for each type of move instruction that might be used by the process.

The *process templates* are instances of *process definitions*, which define the types of move instructions (*move instruction definitions*) that might be used by the process.

To create new move instruction templates, start by creating a new process template for a process that uses the type move instructions you want to create templates for. If such a process does not exist, you first have to create a new process definition.

If no *move instruction definition* for the type of instruction you want to create a template for exists, you must create it first.

When creating instruction descriptions, the virtual controller must be running, since the available instruction types are read from the system.

Method	Description
Creating a process template with move instruction templates on page 97	Move instructions are always related to processes.
Creating a move instruction description on page 98	To create templates for other instructions than the one that already exists in the tree view, you first have to create an instruction description that defines the arguments that belong to the instruction.
Editing an instruction template on page 98	XXX
Creating a process definition on page 99	XXX
Creating an action instruction template on page 99	XXX
Creating an action instruction description on page 99	To create templates for other instructions than the one that already exists in the tree view, you first have to create an instruction description that defines the arguments that belong to the instruction.

Continues on next page

Templates can be imported and exported on four levels: tasks, move instruction descriptions, action instruction descriptions and process definitions. The default directory for imported and exported template files is *My Documents/RobotStudio*. Simply choosing another directory will then make that directory default. The default file format is .xml.

The validation procedure checks for duplicate names, incomplete process definitions and virtual controller equality. It is performed automatically, after a template file has been imported or a node renamed or deleted.

For procedures, see *Instruction Template Manager*.

Instruction Template Manager

The Instruction Template Manager is used to add support for instructions other than the default set that comes with the RobotStudio.

For example, a robot controller system with the RobotWare Dispense option has specialized move instructions related to glueing like DispL and DispC. You can manually define the instruction templates for these using the Instruction Template Manager. The instruction templates are exported to XML format and reused later.

The instruction template supports the following Robotware options:

- Cap (Continuous Application Process)
- Disp (Dispense)
- Trigg (Fixed Position Events)
- Spot Pneumatic
- Spot Servo
- Spot Servo Equalizing
- Paint

RobotStudio has pre-defined XML files that are imported and used for robot controller systems with the appropriate RobotWare options.These XML files has both the Move and Action instructions.

NOTE! It is recommended to use RobotStudio ArcWelding PowerPac while using RobotWare Arc.

Creating a process template with move instruction templates

1. On the **Home** tab, from the active **Task** list, select the task for the robot for which you want to create the instruction template.
2. On the **Create** menu, click **Instruction Template Manager**. This opens the Instruction templates page in the work space.
3. In the **Instruction Templates** tree to the left, make sure there are move instruction definitions for the types of move instructions you want to create templates for. If not, follow the procedure in *Creating a move instruction description on page 98* for creating them.
4. Make sure there is a process definition that uses the types of move instructions you want to create templates for. If not, follow the procedure in *Creating a process definition on page 99* for creating it.
5. Right-click the *process definition* for which you want to create a new templates and click **Create Process Definition**.

Continues on next page

6. In the **Create Process Definition** dialog box, enter a name, with characters from ASCII set, for the new template and click **Create**. A new process template node with a set of move instruction templates is now created.
7. Select each new template one at a time, and in the arguments grid to the right of the tree view, set the argument values that shall be applied when you create new instructions based on the template. Finish for each template by clicking **Apply changes** at the bottom of the grid.

   For details about available arguments and what they do, see the RAPID reference manual for ordinary RAPID instructions and the option manual for software option instructions.

Creating a move instruction description

To create the instruction description, follow these steps:

1. Right-click the **Move Instructions** node and click **Create Move Instruction Description**. This opens the **Create Move Instruction Description** dialog box.
2. In the **Controller Description** list, select the instruction you want to make a description for. All action instructions installed on the controller, both through RobotWare and software options, are available.
3. In the **Motion type** list, select the motion type for the instruction.
4. Optionally, in the **Information text** box, enter a comment to the instruction.
5. Click **Create**. The instruction description appears in the tree view and its settings are displayed in the grid.
6. In the instruction grid, set the **point type** values. If necessary, also change the other settings.
7. After changing the settings, click **Apply Changes** in the bottom of the grid view.

Editing an instruction template

To edit an instruction template, follow these steps:

1. In the active task list, select the task for the robot for which you want to edit the instruction template.
2. On the **Create** menu, click **Instruction Template Manager**. This opens the Instruction templates page in the work space.
3. In the **Instruction Templates** tree to the left, browse to and select the template to edit.
4. In the arguments grid to the right of the tree view, set the argument values that shall be applied when you create new instructions based on the template. Finish by clicking **Apply changes** at the bottom of the grid.

   For details about available arguments and what they do, see the RAPID reference manual for ordinary RAPID instructions and the option manual for software option instructions.

Continues on next page

Creating a process definition

To create a process definition, follow these steps:

1. Make sure there are move instruction definitions for the types of move instructions you want to create templates for. If not, follow the procedure above for creating them.
2. Right-click the **Process definitions** node and click **Create Process Definition**. This opens the Create Process Description dialog box.
3. In the **Create Process Definitions** dialog box, enter a name for the process definition, a name for its first process template and then select the move instruction types to use. Finish by clicking **Create**.

Creating an action instruction template

To create a template for an action instruction, follow these steps:

1. In the active task list, select the task for the robot for which you want to create the instruction template.
2. On the **Home** menu, click **Instruction Template Manager**.
3. In the **Instruction Templates** tree to the left, right-click the instruction description (corresponding to the instructions as described in the RAPID reference manual) for which you want to create a new template and click **Create Action instruction Template**.

   If the instruction description does not exist in the tree, create it by following the procedure described in *Creating an action instruction description on page 99*.
4. In the **Create Action Instruction Template** dialog box, enter a name for the new template and click **Create**. The new template is now created under the instruction description node it belongs to.
5. Select the new template and in the arguments grid to the right of the tree view, set the argument values that shall be applied when you create new instructions based on the template. Finish by clicking **Apply changes** at the bottom of the grid.

   For details about available arguments and what they do, see the RAPID reference manual for ordinary RAPID instructions and the option manual for software option instructions.

Creating an action instruction description

To create the instruction description, follow these steps:

1. Right-click the **Action Instructions** node and click **Create Action Instruction Description**. This opens the **Create Instruction Description** dialog box.
2. In the **Controller Description** list, select the instruction you want to make a description for. All action instructions installed on the controller, both through RobotWare and software options, are available.
3. Optionally, in the **Information text** box, enter a comment to the instruction.
4. Click **Create**. The instruction description appears in the tree view, and its settings are displayed in the grid.

   After changing the settings (if necessary), click **Apply Changes** in the bottom of the grid view.
5. Continue with creating templates for the instruction description, as described in *Creating an action instruction template on page 99*.

3.8. Testing positions and motions

Overview

RobotStudio has several functions for testing how robots reach and move to targets. They are useful both for finding the optimal layout when building a station and during programming.

Below are brief descriptions of the functions for testing reachability and motions.

Checking reachability

The check reachability function displays whether the robot can reach selected targets and motion instructions by changing the frames' colors in the graphic view. Reachable frames are colored green, unreachable red, and frames with reachable positions but not with current orientation are colored yellow.

The reachability check is useful when building the station, since the reachability of several targets are displayed at once. For a procedure, see *Check Reachability on page 325*.

Jumping to target

Jump to target tests whether the robot can reach a specific position. This is useful when building the station: by creating targets at critical positions on the work piece and jumping the robot to them, you get an early indication of whether the items are positioned correctly or not. For a procedure, see *Jump to Target on page 337*.

Viewing a robot at target

When View robot at target is activated, the robot is automatically positioned with the tool at the target when one is selected. If several robot axis configurations are possible for reaching the target, the robot will use the one nearest the configuration it had before jumping to the target. For a procedure, see *View Robot at Target on page 370*.

Viewing tool at target

View tool at target displays the tool at target, without checking that the robot can reach it. This test is useful both when building the station and when programming the robot, since the orientation of targets both affects the reachability and the process performance. For a procedure, see *View Tool at Target on page 371*.

Executing move instructions

Execute move instruction tests if the robot can reach a specific position with the programed motion properties. This is useful for testing motions during programming. For detailed information, see *Execute Move Instruction on page 332*.

Moving along path

Move along path executes all move instructions in a path. It is thereby a more complete test than Execute move instructions, but not as complete as a full simulation, since it ignores RAPID code that is not move instructions. For a procedure, see *Move Along Path on page 354*.

Continues on next page

Moving to a pose

Moving to a pose moves a mechanism to a predefined joint value at a predefined time without using the Virtual Controller. This is useful when movement of the external equipment (such as a clamp or conveyer) must be simulated. For a procedure, see *Move to Pose on page 355*.

Simulating programs

Simulating programs involves running a program on the virtual controller as it is run on a real controller. It is the most complete test whereby you can see how the robot interacts with external equipment through events and I/O signals. For a procedure, see *Simulation Setup on page 257*.

Improving the reachability

If the robot cannot reach the target, or if you are not satisfied with the motions, try the following for improving the reachability:

- Set ConfL or ConfJ to Off for enabling the robot to use new configurations for reaching the target.
- Change the orientation of the target.
- Change the position of either the robot or the work piece.
- Use a system with a track external axis for increasing the robot's range.
- Use a system with a positioner external axis for enabling different work piece positions for different targets.

3.9 Programming MultiMove systems

3.9.1. About programming MultiMove

About MultiMove

The MultiMove functions helps you create and optimize programs for MultiMove systems where one robot or positioner holds the work piece and other robots operate on it. Below is an outline of the main workflow for programming MultiMove systems with RobotStudio, with references to detailed instructions further down in the section.

Prerequisites

For using the MultiMove functions you must first have the following:

- A virtual controller running a MultiMove system started in RobotStudio, see *A MultiMove system with two coordinated robots on page 153* for an example.
- All coordinate systems and tools used by the system.
- The paths along which the tool shall move. The paths must be created in a workobject that belongs to a tool robot and is attached to the work piece robot. A wizard will guide you through attaching the workobjects if this has not been done before starting the MultiMove functions.

For detailed information about MultiMove in RobotWare systems and RAPID programs, see the *MultiMove application manual*.

Normal workflow

This is the typical workflow for creating MutliMove programs using the MultiMove function:

Action	Description
Setting up the MultiMove	Select the robots and paths to use in the program, see *Setting up the MultiMove on page 104*.
Testing the MultiMove	Execute the motion instructions along the paths, see *Testing the MultiMove on page 105*.
Tuning the motion behavior	Tune motion behavior, such as tolerances and constraints for TCP motions, see *Tuning the motion behavior on page 106*.
Creating the program	Generate the tasks for the robots, see *Creating paths on page 108*.

Continues on next page

Additional actions

In addition to using the functions that calculate and create optimized MulitMove paths, you can program MulitMove manaully using a combination of the ordinary programming tools in RobotStudio and a set of tools specific for MultiMove programming.

The main actions for programming MulitMove manually are outlined below. Not all actions might be necessary, but the order in which they shall be carried out depends on the contents of the station and your goals.

Action	Description
Creating Tasklists and Syncidents	This data specifies the tasks and paths that shall be synchronized with each other. See *The Create Tasklist tool on page 212* and *The Create Syncident tool on page 212*, respectively.
Adding and updating ID arguments to the instructions to synchronize	To add IDs to the intructions, you can use one of the following methods: Using *The Recalculate ID tool on page 211* to add and update IDs for instructions in paths that already are synchronized. Using *The Convert path to MultiMove path tool on page 212* to add IDs to instructions in paths that have not yet been synchronized.
Adding and adjusting Sync instructions to the paths.	Add `SyncMoveon/Off` or `WaitSyncTask` instructions to the paths to synchronize and set their tasklist and Syncident parameters. See *Creating an action instruction on page 215*.
Teaching MultiMove instructions	It is also possible to jog all robots to the desired positions and then teach instructions to new synchronized paths. See *MultiTeach tab on page 208*.

3.9.2. Setting up the MultiMove

Selecting robots and paths

This procedure is for selecting the robots and paths in the station that shall be used for the MutliMove program. All robots for the MultiMove program must belong to the same system.

1. On the **Home** tab, click **MultiMove**. Click the **Setup** tab below the MutliMove work area.
2. In the work area, click the **System config** bar for expanding the system config section.
3. In the **Select System** box, select the system that contains the robots to program.

   The robots of the selected system are now displayed in the System grid below the Select system box.
4. For each robot that shall be used in the program, select the check box in the **Enable** column.
5. For each robot that shall be used in the program, specify whether it carries the tool or the work piece using the options in the **Carrier** column.
6. In the work area, click the **Path config** bar for expanding the path config section.
7. Select the **Enable** check box for the tool robot and click the expand button. This displays the paths of the robot.
8. Select the order of the paths to execute by specifying them in right order using **Path name** column.
9. For each path that shall be included in the program, select the check box in the **Enable** column.
10. When you have set up the robots and paths, continue testing the Multimove and then tune the motion properties, if necessary.

3.9.3. Testing the MultiMove

Overview

Testing the MultiMove executes the motion instructions along the paths according to the current settings on the setup and motions properties pages.

Testing Paths

This procedure is for setting the robots start position and testing the resulting movements along the path sequence.

1. Jog the robots to what seems to be a good start position.
2. On the **Home** tab, click **MultiMove**. Click the **Test** tab at the bottom of the MultiMove work area for displaying the test area.
3. Optionally, select the **Stop at end** check box to make the simulation stop after moving along the paths. Clearing this check box makes the simulation continue in a loop until you click **Pause**.
4. Click **Play** to simulate the motions along the paths based on the current start position.

   If you are satisfied with the motions, continue generating multimove paths. If the simulation cannot complete or if you are not satisfied with the motions, pause the simulation and perform any of the actions below to adjust the motions:

Action	Description
Examine the robots' positions for critical targets.	Click **Pause** and then use the arrow buttons to move to one target a time.
Jog the robots to new start positions.	New start positions might result in changed motions, since the robots will use different configurations. In most cases, positions near the robots' joint limits shall be avoided.
Go to the **Motion Behavior** tab and remove constraints.	The default setting for the motion properties is no constraints. If this has been changed, constraints might exist that limit motions more than necessary.

3.9.4. Tuning the motion behavior

Overview

Tuning the motion behavior means to set up rules for the robot's motions, for example, constraints on the position or orientation of the tool. Generally, the MultiMove program will obtain the smoothest motions with the fastest cycle and process times with as few constraints as possible.

For procedures, see *Motion Behavior tab on page 205*.

Modifying the joint influences

The joint influence controls the balance of how much the robots will use their joints. Decreasing the weight value for one axis will restrict the motion for this axis, while increasing it will promote motion on this axis relative to alternative axes.

1. On the **Home** tab, click the **Motion Behavior** tab.
2. Expand the **Joint Influence** group by clicking its title bar.
3. In the Select Robot box, select the robot whose joint influence you want to modify.

   The weight values for the robot axes are now displayed in the grid.
4. For each axis whose motion you want to restric or promote, adjust the Weight value. A lower value restricts, and a higher value promotes, motions on that axis.

Modifying the TCP constraints

The joint influence controls the balance of how much the robots will use their joints. Decreasing the weight value for one axis will restrict the motion for this axis, while increasing it will promote motion on this axis relative to alternative axes.

1. On the **Simulation** tab, click the **Motion Behavior** tab.
2. Expand the **TCP Constraints** group by clicking its title bar.

   The directions and rotations in which you can constrain the TCP's motion are now displayed in the grid
3. For each pose you want to constrain, select the **Enable** check box and specify the constraint values (location in the TCP coordinate system). To use the values from the current TCP position, click **Pick from TCP**.
4. Optionally, adjust the **Weight** value for the constraint. A low value results in a harder constraint, while a high value allows a larger deviation.

Modifying the tool tolerance

The joint influence controls the balance of how much the robots will use their joints. Decreasing the weight value for one axis will restrict the motion for this axis, while increasing it will promote motion on this axis relative to alternative axes.

1. On the **Simulation** tab, click the **Motion Behavior** tab.
2. Expand the **Tool Tolerance** group by clicking its title bar.

   The directions and rotations in which you can enable tolerances are now displayed in the grid
3. For each offset you want to set, select the **Enable** check box.

Continues on next page

4. In the **Value** column, specify the allowed deviation.
5. Optionally, adjust the **Weight** value for the tolerance. A low value increases the use of the tolerance, while a high value promotes motions that do not use the tolerance.

Modifying the tool offset

The tool offset sets a fixed distance between the tool and the paths.

1. On the **Simulation** tab, click the **Motion Behavior** tab.
2. Expand the **Tool Offset** group by clicking its title bar.

   The directions and rotations in which you can set offsets are now displayed in the grid.
3. For each offset you want to set, select the **Enable** check box.
4. In the **Offset** column, specify the offset distance.

3.9.5. Creating paths

Overview

When you are satisfied with the motions displayed when testing the Multimove program, the next step is to convert the temporary move instructions used by the MultiMove function to ordinary paths in RobotStudio.

Creating the paths

To create paths for the MultiMove program in RobotStudio, follow these steps:

1. On the **Home** tab, click **Create Paths** tab.
2. Expand the **Settings** group by clicking on the its title bar.
3. Optionally, change the naming settings in the following boxes:

Box	Description
Start ID	Specify the first ID number for the synchronization of the instructions for the robots.
ID step index	Specify the increment between ID numbers.
Sync ident prefix	Specify a prefix for the syncident variable, which connects the sync instructions in the tasks for the tool robot and the work piece robot with each other.
Task list prefix	Specify a prefix for the tasklist variable, which identifies the tasks for the tool robot and the work piece robot to synchronize.

4. Expand the **WP Robot Settings** group by clicking on its title bar and then check the settings in the following boxes:

Box	Description
WP Workobject	Specify the workobject to which the targets generated for the workpiece robot shall belong.
WP TCP	Specify which tooldata the workpiece shall use when reaching its targets.
Path prefix	Specify a prefix for the generated paths.
Target prefix	Specify a prefix for the generated targets.

5. Expand the **Generate path** group by clicking on its title bar and then click **Create Paths**.

3.9.6. Programming external axes

Overview

This is a brief overview of the functions and commands for programming external axes in RobotStudio. For a more detailed description of external axes and how to program them, see the product manual for the external axis to use and the *RAPID reference manual*.

Coordinated motions

Normally, external axes are used to move the workpiece, the robot or any other mechanism. The motions of an external axis can be coordinated with those of a robot in two ways, depending on the task in which the external axis is defined.

Task for external axis	Coordination method
Same task as the robot´s	If the external axis is in the same task as the robot, the current position of active external axes is stored with each target that is created. When the robot then moves to the target, the external axis will move to the stored position as well. Modifying and optimizing the position of positioner external axes can be automated using the MultiMove function, or be performed manually for selected targets. Positions of track external axes can only be modifed manually. For information about using the MultiMove function, see *About programming MultiMove on page 102*. For information about how to modify the position of external axes manually, see below.
Other task than the robot's	If the external axis is in another task than that of the robot it shall be coordinated with, the motions of the external axis are created by *MoveExt* instructions, and the coordination is made by *sync* instructions. For positioner external axes creating or optimizing MoveExt and sync instructions can be automated way using the MultiMove function, or be performed manually by creating a path with MoveExt instructions for the positioner and then adding sync instructions to the path for the robot and the external axis. Track external axes can only be programed manually. For information about using the MultiMove function see *About programming MultiMove on page 102*. For information about how to use the sync instructions, see the *RAPID reference manual* and the *MultiMove application manual*.

Continues on next page

Modifying positions of external axes

When programming external axes, you often need to adjust the position of the external axis for some targets. For example, if you create a path from curves on a work piece that is attached to a positioner, the positioner will initially have the same position for all targets. By repositioning the work piece for some of the targets you might improve process time and reachability.

When targets are created in stations with a coordinated external axis, the position values of the external axis are stored in the target. With the Modify External Axis function you can reposition the external axis, thus making it possible for the robot to reach the target in new ways. For a procedure, see *Modify External Axis on page 349*.

To modify the external axis values for a target, the following conditions must be met:

- The external axis must be added to the system and set up correctly. For examples of how to add support for an external axis to a system, see *A system with support for one robot and one track external axis* and *A system with support for one robot and one positioner external axis on page 155*. For information about how to set up an external axis in a RobotStudio station, see *Placing external axes on page 79*.
- The external axis must be defined in the same task as the robot.
- The external axis must be activated.

Activation and deactivation

Activating a mechanical unit makes it controlled and monitored by the controller. Consequently, the mechanical unit must be activated before programming or running programs. If a system uses several external axes or interchangeable models with several work stations, several mechanical units might share common drive units. If this is the case, you must make sure to set the mechanical unit as active.

For more information about activating and deactivating mechanical units, see the *RAPID reference manual* on the instructions *ActUnit* and *DeactUnit*.

Activating and deactivating mechanical units can be done either manually, see *Activate Mechanical Units on page 264*, or programmatically by RAPID instructions, see below.

To activate or deactivate mechanical units programmatically

To set the mechanical units to be active programmatically by RAPID instructions, follow these steps:

1. In the **Paths&Targets** browser, browse down to the path in which you want to insert the activation or deactivation instruction. To insert it as the first instruction in the path, select the path node and to insert it between existing instructions, select the instruction before the intended insertion point.
2. On the **Home** tab, click **Action Instruction** to bring up a dialog box.
3. In the **Instruction Templates** list, select one of the **ActUnit** or **DeactUnit** instructions.
4. In the **Instruction Arguments** grid and the **MechUnit** list, select the unit to activate or deactivate.
5. Click **Create**. When the path is executed either through the Move along path command, or running the RAPID program, the instruction will be carried out.

3.10. Loading and saving programs and modules

Overview

RAPID programs and modules are normally stored in the RobotWare systems, as they are created. You can also save the programs to files on the PC, which makes it possible to load them to other controllers, either other virtual controllers or real IRC5 controllers.

Programs are saved from the VC

When saving a program to files on the PC from RobotStudio, it is the RAPID program stored in the system of the VC that is saved. This program is created and updated by synchronizing the station to the VC, see *Synchronize to VC on page 275*.

Procedures

To create or load a module or load a program, see:

- *New Module on page 289*.
- *Load Module on page 290*.
- *Load Program on page 291*.

To save a module or program, see:

- *Save Module As on page 363*.
- *Save Program As on page 364*.

3.11. Synchronization

Overview

To synchronize is to make sure that the RAPID program in the system running on the virtual controller corresponds to the programs in RobotStudio. You can synchronize both from RobotStudio to the virtual controller and from the virtual controller to RobotStudio.

In a RobotStudio station, robot positions and movements are defined by targets and move instructions in paths. These correspond to data declarations and RAPID instructions in the modules of the RAPID program. By synchronizing the station to the virtual controller, you create RAPID code out of the data in the station. By synchronizing the virtual controller to the station, you create paths and targets out of the RAPID program in the system running on the virtual controller.

When to synchronize the station to the VC

Synchronizing the station to the VC updates the RAPID program of the virtual controller with the latest changes in the station. This is useful to do before:

- Performing a simulation.
- Saving a program to files on the PC.
- Copying or loading RobotWare systems.

To synchronize a statiuon to the VC, see *Synchronize to VC on page 275*.

When to synchronize the VC to the station

Synchronizing the VC to the station creates paths, targets and instructions that correspond to the RAPID program in the system running on the virtual controller. This is useful to do when you have:

- Started a new virtual controller which system contains existing programs.
- Loaded a program from a file.
- Text-edited the program.

To synchronize the VC to a station, see *Synchronize to Station on page 274*.

3.12. Using the RAPID editor

Overview

The integrated RAPID program editor is useful for editing all robot tasks other than robot motion. This section provides instructions on launching the editor and case examples illustrating several useful functions, such as keyboard shortcuts, IntelliSense, code snippets and the watch window.

Opening the program editor

To access the program editor, perform the following steps:

1. Click the **Offline** tab.
2. In the **Program** browser, select a module for editing and then click **RAPID Editor.**

   The RAPID code of the program appears as a tabbed document window.

TIP!

The graphical layout can be consulted without closing the editor by clicking the graphics window tab.

Searching: a case example

Assume that you have programed targets and motion instructions and sychronized them to the controller. The number of targets is large, so you decide to distribute them among several modules.

You may have forgotten in which module your main procedure is found.

1. Press **CTRL + F** to bring up the **Search and Replace** dialog box.
2. In the **Search Pattern** box, type "PROC main". Since no modules are open, in the **Search In** list, select **Current System**, and then click **Search All**.

   The search result is displayed in the output window.
3. Double-click the line matching your search to launch the program editor.

   The entire RAPID program is then checked for errors.

Continues on next page

Code Snippets

Code Snippets are pieces of code inserted in the RAPID Editor. The editor integrates these codes with the pick list. The following are the predefined code snippets in RobotStudio:

- array2x2x4.snippet
- array2x4.snippet
- array2x4x2.snippet
- array4x2.snippet
- function with return value bool.snippet
- module header.snippet
- procedure parameters.snippet
- procedure with error handler.snippet
- robtarget.snippet
- tooldata.snippet
- TRAP routine example.snippet
- wobjdata.snippet

You can create customized code snippets and add to the existing list.

NOTE!

Code Snippets can be edited in an XML editor such as Visual Studio 2005 Express. For more information about creating customized code snippets, see *http://msdn.microsoft.com/en-us/library/ms165394.asp*.

Editing: a case example

Assume that you wish to create an infinite loop whereby the controller receives commands from a line PLC. The controller communicates with the PLC using digital I/O signals, but you have forgotten the exact name of the function that reads an input signal.

1. Using code snippets, create a new procedure.
2. Press **CTRL + SHIFT + SPACE** to open the pick list.
3. Double-click the **I/O** folder, and then double-click the **DOutput** instruction to insert it at the insertion position.
4. Press the spacebar to display the parameter information ToolTip. As you enter parameters, the ToolTip is updated, displaying the current argument in bold. The ToolTip is closed either by concluding the instruction with a semicolon (;), or by pressing **ESC**.

Continues on next page

TIP!

At any time you may press **CTRL + SHIFT** to complete what you have begun typing. This will either bring up a narrowed-down list of selectable parameters, or, if only one selection remains, will automatically complete your text.

Adding breakpoints: a case example

Now that you have finished editing, you may want to test your loop and add some breakpoints.

1. Place the insertion on the new statement and press **F9** to set a breakpoint.

   The breakpoint, indicated by a grey circle, is stored in the editor until changes are applied, whereupon it turns red.
2. Ensure that the **Ignore breakpoints** button in the **Program editor** toolbar is not clicked, and click the **Play** button on the **Simulation** toolbar.

   The program will run and then stop at the breakpoint.
3. To run the program statement by statement, click the **Step over** button in the **Program editor** toolbar.

Applying and verifying the edits

To apply and verify your edits, follow these steps:

1. To apply your edits, click the **Apply** icon in the editor toolbar.

   The statement is highlighted, and breakpoint locations are adjusted, if necessary.
2. To verify the syntactic and semantic correctness of the modules, click the **Check Program** icon in the editor toolbar.

Executing: a case example

You might want to debug your loop or monitor a specific variable.

1. In the program editor browser, right-click the procedure you want to set as entry point, and then click **Set Entry**.

   The icon is marked in red.
2. In the **Simulation** tab, click the **Play** button.

   The program will run and then stop at the next breakpoint.
3. On the **View** menu, click **Watch Window** to bring up the watch window.
4. Select a variable for monitoring and drag it to the watch window.
5. Restart the loop and monitor the variable at each iteration.

4 How to simulate programs

4.1. Simulation Overview

About this chapter

This chapter describes how to simulate and validate robot programs. Below are short introductions to the simulation functions in RobotStudio.

Function	Description
Play simulations	Simulations run entire robot programs on a virtual controller. Before you run a simulation you need to decide which paths are to be simulated. To set up a simulation, see *Simulation Setup on page 257*. To run a simulation, see *Simulation Control on page 265*.
Collision detection	Collision detection displays and logs collisions and near-misses for specified objects in the station. Normally used during simulation of robot programs, it can also be used when building the station. For more information, see *Detecting collisions on page 119*.
Event handling	Events can be used to connect an action to a trigger. For example, you can attach one object to another when they collide or a signal is set. For more information, see *Creating an event on page 122*.
I/O Simulation	In simulations I/O signals are normally set either by the robot program or by events. With the I/O simulator you can set signals manually, which provides a quick test of specific conditions. For more information, see *Simulating I/O signals on page 123*.
Simulation Monitoring	With the simulation monitoring functions you enhance the simulation by adding traces along the TCP movements or alerts triggered by defined speeds or motions. For more information, see *Enabling simulation monitoring on page 124*.
Process time measurement	With the process timer you measure the time for a process to complete. For more information, see *Measuring process time on page 125*.

Time handling during simulation

When simulating stations with events or several controllers, or other time managing equipment, time can be managed in two modes: either as *free runtime* or as *time slices*. RobotStudio uses time slice mode by default, but you can switch to free runtime, if required.

Free runtime

Since all controllers use the same computer resources, their synchronization might not be exactly as in the real world if they run independently of each other (called *free run* mode). The cycle time will be correct, but the timing for setting signals and triggering events might be inaccurate.

Time Slice

Time slices can be used to ensure that the timing for signals and other interaction between controllers is accurate. In this mode, RobotStudio synchronizes the controllers by dividing a time segment into small slices and waiting for all controllers to complete a current time slice before any controller can start anew. Thus, the controllers are synchronized, and the cycle

Continues on next page

time will be calculated correctly. The drawbacks are that the virtual FlexPendant cannot be open, and that the simulation might be somewhat slow and jerky, depending on the complexity of the simulation and the performance of the computer.

NOTE!

If the simulation uses events or involves several different controllers, the virtual time mode **Time Slice** shall be used to make sure that the timing between the controllers is correctly simulated.

4.2. Detecting collisions

Overview

With RobotStudio you can detect and log collisions between objects in the station. The basic concepts of collision detection are explained below.

Collision sets

A collision set contains two groups, *Objects A* and *Objects B*, in which you place the objects to detect any collisions between them. When any object in *Objects A* collides with any object in *Objects B*, the collision is displayed in the graphical view and logged in the output window. You can have several collision sets in the station, but each collision set can only contain two groups.

A common use of collision sets is to create one collision set for each robot in the station. For each collision set you then put the robot and its tool in one group and all objects you do not want it to collide with in the other. If a robot has several tools, or holds other objects, you can either add these to the robot's group as well or create specific collision sets for these setups.

Each collision set can be activated and deactivated separately.

Collisions and near-misses

In addition to collisions, the collision detection can also watch for near-misses, which is when an object in *Objects A* comes within a specified distance from an object in O*bjects B*.

Recommendations for collision detection

In general, the following principles are recommended to facilitate collision detection:

- Simplify and defeature your models by removing everything that is not necessary for simulation purposes.
- Use as small collision sets as possible, splitting large parts and collecting in the collision sets only relevant parts.
- Enable coarse detail level when importing geometry.
- Limit the use of near-miss.
- Enable last collision detection, if the results are acceptable.

Results of creating a collision set

After you have created a collision set, see *Create Collision Set on page 256*, RobotStudio will check the positions of all objects and detect when any object in *ObjectsA* collides with any object in *ObjectsB*.

Activation of detection and displayal of collisions depend on how the collision detection is set up.

If the collision set is active, RobotStudio will check the positions of the objects in the groups, and indicate any collision between them according to the current color settings.

Continues on next page

Collision detection

Collision detection checks whether robots or other moving parts collide with equipment in the station. In complex stations, you can use several collision sets for detecting collisions between several groups of objects.

After collision detection has been set up, it does not need to be started, but automatically detects collisions according to the setup.

Setting when to check for collisions

To set whether to detect collisions always or only during simulation, follow these steps:

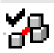
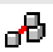
1. On the **Application** menu, click **RobotStudio Options** and under **Simulation**, click **Collision**.
2. In the **Perform collision detection** option, select one of the following:

Option	Description
During simulation	Collision detection is active only during simulation (when running RAPID programs in the virtual controller).
Always	Collision detection is always active, even when moving objects manually or testing reachability.

Setting the objects for collision detection

To set the objects for collision detection, follow these steps:

1. Make sure that the objects for collision detection are placed correctly in collision sets.
2. Make sure that the collision set for the objects is activated, which is indicated by an icon in the Layout browser:

Icon	Description
 xx0500001552	Active. Collisions between objects in this set will be detected.
 xx0500001370	Not active. Collisions between objects in this set will not be detected.

To activate or deactivate collision sets, continue with the following steps:

3. Right-click the collision set to change and then click **Modify Collision set** to bring up a dialog box.
4. Select or clear the **Active** check box and then click **Apply**.

Setting near-miss detection

Near-misses occur when objects in collision sets are close to colliding. Each collision set has its own near-miss settings. For setting near-miss detection, follow these steps:

1. In the **Layout** browser, right-click the collision set to change and then click **Modify Collision set** to bring up a dialog box.
2. In the **Near miss** box, specify the maximum distance between the objects to be considered a near-miss and then click **Apply**.

Continues on next page

Setting logging options

In addition to the graphical display of collisions, you can also log the collisions to the output window or a separate log file:

1. On the **Application** menu, click **RobotStudio Options** and under **Simulation**, click **Collision**.
2. For displaying the collision log in the output window, select the **Log collisions to Output window** check box.
3. For logging collisions to a separate file, select the **Log collisions to file** check box and enter the name and path to the log file in the box below the check box.

4.3. Creating an event

Overview

Events enhance your simulations by defining actions that are carried out when specific trigger conditions are fulfilled. You can use events to:

- Attach one object to another, for example, a work piece to a gripper when simulating material handling, see *Attaching and detaching objects on page 48*.
- Set signals, for example, when simulating signals set by equipment other than the controller, see *Simulating I/O signals on page 123*.
- Start or stop the process timer, see *Measuring process time on page 125*.

Used for creating new events, the **Create New Event Wizard** is launched from the Event manager, see *Event Manager on page 258*.

Prerequisites

Before creating the event, make sure that the station contains all signals and objects that are planned to be used as triggers or affected by the action.

4.4. Simulating I/O signals

Procedures

When simulating I/O signals you can either create events that set signal values when specified trigger conditions are fulfilled, or you can set signal values manually.

For procedures using the event manager, see *Event Manager on page 258*.

For procedures using the I/O simulator, see *I/O Simulator on page 266*.

Related information

For information about controlling I/O signals from the RAPID program, see *Creating RAPID instructions for setting I/O signals on page 95*.

4.5. Enabling simulation monitoring

Overview

The simulation monitor commands are used to visually detect critical robot movements during simulation by drawing a colored line that follows the TCP.

To enable TCP tracing

To enable TCP tracing, follow these steps:

1. On the **Simulation** tab, click **Monitor** to bring up a dialog box.
2. In the left pane, select the appropriate robot.
3. On the **TCP Trace** tab, select the **Enable TCP Trace** check box. This activates TCP tracing for the selected robot.
4. Optionally, change the length and color of the trace. For detailed information, see *Monitor on page 268*.

To enable simulation alerts

To enable simulation alerts, follow these steps:

1. On the **Simulation** menu, click **Monitor** to bring up a dialog box.
2. In the left pane, select the appropriate robot.
3. On the **Alerts** tab, select the **Enable Simulation Alerts** check box. This activates simulation alerts for the selected robot.
4. In the threshold value boxes, specify the threshold for the alerts. Setting the threshold to 0 is equivalent to disabling the alert. For detailed information, see *Monitor on page 268*.

4.6. Measuring process time

Overview

The process timer measures the time it takes for robots to run a simulation or move along a path. When the timer is activated, the clock will start as soon as a process is started. If two processes are started at the same time, the timer will not stop until the last one has stopped.

To measure the process time

To measure the process time, follow these steps:

1. On the **Simulation** tab, click the **Clock** button.
2. Play the simulation or move along path.

   The timer stops when the last process has stopped. If you start another process without having deactivated the **Clock** button, the timer will continue.
3. To reset the clock to zero, click the **Reset Process Timer**.

5 Deployment and distribution

5.1. Copying programs

Overview

RAPID programs are normally stored in the systems that run on the virtual controllers of your station. To copy programs to systems on other controllers, save the programs to file on the PC and then load these files to the destination controllers. You can save either entire programs or specific modules.

Copying a program

To copy a program from one controller to another, follow these steps:

1. In the **Offline** browser, select the controller that contains the program to copy.
2. Save the program to file on the disc. For details, see *Saving a program on page 364*.
3. If necessary, copy the files to a location that is accessible to the other controller.
4. For instructions on loading the program to a system on a virtual controller, a FlexController or a non-running system, see the table below.

System location	Do this
Virtual controller, running in RobotStudio	See *Saving a module on page 363*.
FlexController	Connect to the FlexController and load the program.
A non-running system stored on the PC	Start the system in a virtual controller, then load the program, see *Adding a system on page 65* and *Saving a module on page 363*, respectively.

5.2. Pack & Go / Unpack & Work

Overview

The Pack & Go / Unpack & Work feature makes it possible to create a package (zip file) of an active station that can be unpacked on another computer. The package contains all necessary files, except media pools, but additional, option-based media pools are included.

For procedures, see *Pack & Go on page 174* and *Unpack & Work on page 175*.

5.3. Screen Capture

Overview

Screen capture entails two functions useful for demonstrations and training purposes:

- The screenshot function allows you to capture an image of the application, see *Screenshot on page 173*.
- The recorder allows you to make a recording of your work in RobotStudio, either of the entire GUI or just the graphics window, see *Record Movie on page 269*.

Prerequisites

Recording a movie requires that Windows Media Encoder (WME) 9 is installed on your computer.

6 Working online

6.1. Connecting a PC to the service port

NOTE!

The service port shall only be used for direct connection to a PC as described in this procedure. It must not be connected to a LAN (local area network), since it has a DHCP server that automatically distributes IP addresses to all units connected to the LAN.

Contact your network administrator for more information.

NOTE!

The maximum number of connected network clients using robapi is:

- LAN: 3
- Service: 1
- FlexPendant: 1

Continues on next page

The total max. number of applications using robapi running on the same PC connected to one controller has no built-in maximum however, UAS limits the number of logged on users to 50.

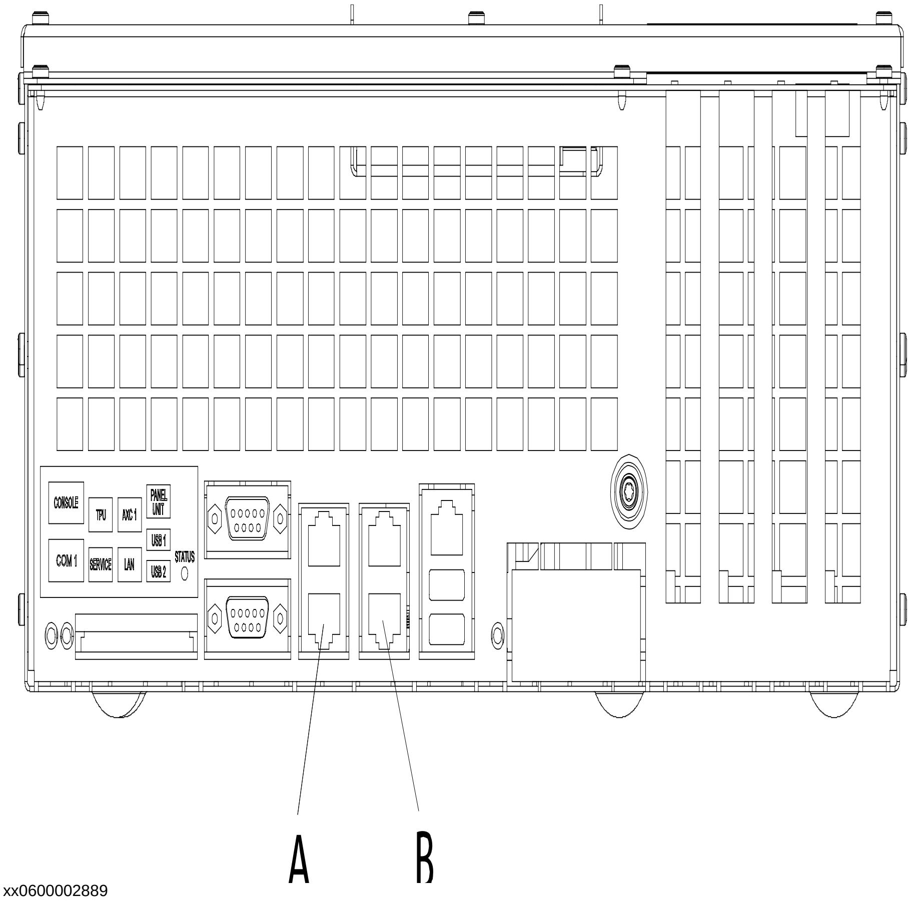
The total max. number of concurrently connected FTP clients is 4.

CAUTION!

When a cable is connected to the service port on the controller front and the service hatch or the connector cover on the single cabinet is opened, the controller will not comply with the requirements of protection class IP54.

Ports DSQC639

The illustration below shows the two main ports on the computer unit: the Service Port and the LAN port. Make sure the LAN (factory network) is not connected to any of the service ports!



xx0600002889

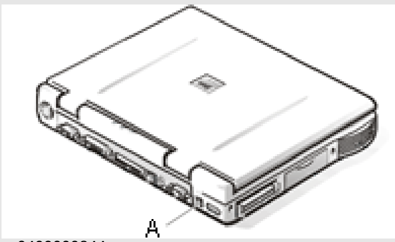
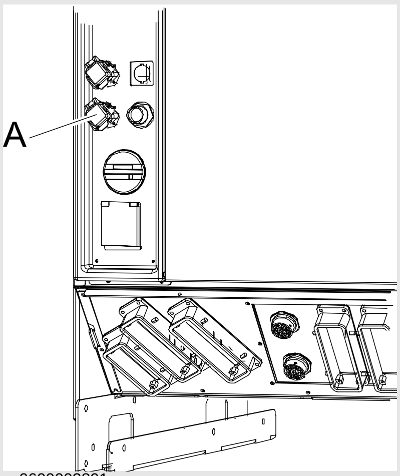
A	Service port on the computer unit (connected to Service port on the Control Module front through a cable) Service port on the computer unit
B	LAN port on computer unit (connects to factory LAN)

Continues on next page

Connections to ports may be done as detailed below.

Connection to/from:	Detailed in section:
Connecting the Single Cabinet/Control Module to the factory LAN.	
Connecting a PC to the Single Cabinet/ Control Module service port.	Proceed as detailed below.

Connecting a PC to the service port

	Action	Illustration
1.	Make sure the network setting on the PC to be connected is correct.	Refer to the system documentation for your PC, depending on the operative system you are running. The PC must be set to “Obtain an IP address automatically” or set as described in **Service PC Information** in the Boot Application.
2.	Use the delivered category 5 Ethernet crossover boot cable with RJ45 connectors.	The cable is delivered in the RobotWare product box.
3.	Connect the boot cable to the network port of your PC.	 xx0400000844 • A: network port The placement of the network port may vary depending on the PC model.
4.	Connect the boot cable to the service port on the controller. Connect the boot cable to the service port on the computer unit.	 xx0600002891 • A: Service port

6.2. Network settings

Overview

This topic describes the network settings for a PC connected to a controller, which is a prerequisite for working online.

You can connect the PC to the controller through an Ethernet network in the following ways:

- Local network connection
- Service port connection
- Remote network connection

Local network connection

You can connect your PC to the same Ethernet network that the controller is connected to. When the PC and the controller are connected correctly and to the same subnet, the controller will be automatically detected by RobotStudio.

The network settings for the PC depend on the network configuration. For setting up the PC, contact the network administrator.

Service port connection

When connecting to the controller's service port, you can either obtain an IP address for the PC automatically, or you can specify a fixed IP address.

If you are not sure how to set up the service port connection, contact the network administrator.

Automatic IP address

The controller's service port has a DHCP server that will automatically give your PC an IP address if it is configured for this. For detailed information see Windows help on configuring TCP/IP.

Fixed IP address

Instead of obtaining an IP address automatically, you can also specify a fixed IP address on the PC you connect to the controller.

Use the following settings for a fixed IP address:

Property	Value
IP address	192.168.125.2
Subnet mask	255.255.255.0

For detailed information about how to set up the PC network connection, see Windows help on configuring TCP/IP.

Continues on next page

NOTE!

Obtaining an IP address automatically might fail if the PC already has an IP address from another controller or Ethernet device.

To ensure that you get a correct IP address if the PC was previously connected to an Ethernet device, do one of the following:

- Restart the PC before connecting to the controller.
- Run the command `ipconfig /renew` from the command prompt after connecting the PC to the controller.

Remote network connection

To enable a connection to a controller on a remote subnet, the relevant network traffic must be allowed through any firewalls between the PC and the controller.

The firewalls must be configured to accept the following TCP/IP traffic from the PC to the controller:

- UDP port 5514 (unicast)
- TCP port 5515
- Passive FTP

All TCP and UPD connections to remote controllers are initiated by the PC, that is the controller only responds on the given source port and address.

Connecting to the controller

1. Make sure the PC is connected to the controller's service port and that the controller is running.
2. On the **Application** menu, point to **Online** and then click **One Click Connect**.
3. Select the controller in the **Online** tab
4. Click **Request Write access**.

If the controller is in mode	Then
Auto	You will now get Write Access if it is available.
Manual	A message box on the FlexPendant will allow you to grant remote Write Access to RobotStudio Online.

6.3. User Authorization

Overview

This section describes the controller's User Authorization System (UAS), which restricts what different users are allowed to do with the robot. This is for protecting data and functionality from unauthorized use.

The user authorization is managed by the controller, which means that the UAS settings remain for the controller regardless of which system it is running. It also means that the UAS settings apply to all tools for communicating with the controller, like RobotStudio Online or the FlexPendant. The UAS settings defines the users and groups that can access the controller, and what actions they are granted access to.

For procedures, see *User Accounts on page 306*.

Users

In UAS users are accounts with which persons log on to the controller. Furthermore, the users are added to groups to which access grants are given.

The users are defined in the controller by a user name and a password. For logging on to a controller, the user must type in a defined user name with a correct password.

A user can either have the state activated or deactivated in the UAS. When a user is deactivated it is not possible to log on to the controller using that account. It is the UAS administrator that activates and deactivates the users.

The Default user

All controllers have a default user named *Default User*, which does not have any password and cannot be removed. Other users can be added and removed by the user having the grant *Manage UAS settings*.

Groups

In UAS, groups are defined sets of grants for accessing the controller. To the groups you then add the users who shall have the grants defined by the group.

A good practice is to create groups that resembles the professions that work with the robots in your organization. For example, you can create groups for administrators, programmers and operators.

The Default group

All controllers have a default group named *Default Group*, to which all grants are given and to which the default user belongs. This group cannot be removed, but it can be changed by the user having the grant *Manage UAS settings*.

Continues on next page

Continued

NOTE!

There is a risk changing the group membership of the default user. If you by mistake clear the *Default User* check box or any *Default Group* grant, you will get a warning. Make sure that there is at least one user defined that has the grant *Manage UAS settings*. If the *Default group* and no other group have the grant *Manage UAS settings*, you may lose your ability to manage users and groups.

Grants

Grants are permissions to perform actions or access data on the controller. You use the grants by giving them to groups, to which you then add the users who shall have the grants.

Grants may be either *controller grants* or *application grants*. Depending on the actions that you will perform, you may need several grants. For procedures, see *UAS Grant Viewer on page 311*.

Controller grants

Controller grants are validated by the robot controller and apply to all tools and devices accessing the controller.

Application grants

Application grants are used by a specific application, for example the FlexPendant, and will only be valid using that application. Application grants can be added by additional options and used in customer applications.

6.4 The System Builder

6.4.1. System Builder Overview

Overview

This section describes how you create, build, modify and copy systems to run on virtual and real controllers. These systems may even be converted to boot media and downloaded to a real controlller.

The system points out the robot models and options to use; it also stores configurations and programs for the robots. Therefore, it is good practice to use a unique system for each station even if the stations use the same basic setup. Otherwise, changes in one station may accidently overwrite data used in another station.

About virtual and real systems

The system you run on virtual controllers can either be a real system built on real RobotWare keys or a virtual system built on virtual keys.

When using real systems, the RobotWare keys define which options and robot models shall be used, thus helping you to configure the system correctly. Real systems can be run both on virtual controllers and real IRC5 controllers.

When using virtual keys, all options and robot models are available, which is useful for evaluation purposes, but requires more configuration when creating the system. Systems built on virtual keys can only be run on virtual controllers.

Prerequisites

Creating a system entails applying a predefined template to a station, reusing an existing system or letting RobotStudio propose a system based on a layout.

To create a system, the following conditions must be met:

- The RobotWare media pool must be installed on your PC.
- You must have a RobotWare key for the system, if creating a system to run on a real controller. The RobotWare key is a license key that determines which robot models to use and which RobotWare options to run on the controller. The license key is delivered with the controller.
- If you want to create a system for virtual use only, you can use a virtual key instead. Virtual keys are generated by the wizard. When using virtual keys, you select the robot models and options to use in the *Modify Options* section of the wizard.
- Downloading to the real controller requires a direct connection from your computer to the service or Ethernet port of the controller.

Continues on next page

Administering systems

Systems can be administered from the **System Builder** dialog box in the following ways:

- View system properties, see *Viewing system properties on page 141*.
- Build a system, see *Building a new system on page 142*.
- Modify or delete a system, see *Modifying a system on page 145*.
- Copy a system, see *Copying a system on page 149*.
- Create a system from backup, see *Creating a system from backup on page 150*.
- Download a system to a controller, see *Downloading a system to a controller on page 151*.
- Create boot media, see *Creating boot media on page 152*.

6.4.2. Viewing system properties

Overview

All systems you create with the System Builder are stored locally on your computer. It is recommended that you store them in one or more dedicated system directories.

Viewing system properties

To view system properties and add comments, follow these steps:

1. In the **System Builder** dialog box, select a system from the **Systems** box.

   If necessary, in the **System directory** list, you can navigate to the folder in which your systems are stored
2. The system properties are then displayed in the **System Properties** box. Optionally, type a comment in the **Comments** box, and click **Save**.

6.4.3. Building a new system

Overview

The **New Controller System Wizard**, used for building a new system, is launched from the System Builder.

Starting the wizard

To start the wizard, follow these steps:

1. Click **System Builder** to bring up a dialog box.
2. In the **Actions** group, click **Create New**. This starts the wizard.
3. Read the information on the welcome page and click **Next**.

Specifying the name and location

To determine where on your computer to store the system you are creating, follow these steps:

1. In the **Name** box, enter a name for the system you are creating.
2. In the **Path** box, enter the path to the system directory in which you will store the system.

   You can also click the **Browse** button and browse to the system directory.
3. Click **Next**.

Entering the RobotWare keys

The RobotWare keys determine which RobotWare versions and parts to use in the system.

Creating a system to run on either IRC5 controllers or virtual controllers requires at least two keys: one for the controller module and one for each drive module in the cabinet. The keys are delivered together with the controller.

For creating a system to run on virtual controller only (for example, in Virtual IRC5), you can use virtual keys. Virtual keys give access to all options and robot models, but limits the use of the system to virtual controllers only.

To enter the key for the controller module, follow these steps:

1. In the **Controller Key** box, enter the controller key. You can also click **Browse** and browse to the key file. If creating a system for virtual use only, select the **Virtual Key** check box, and the controller key will be generated by the wizard.
2. In the **Media Pool** box, enter the path to the media pool. You can also click **Browse** and browse to the folder.system
3. In the **RobotWare Version** list, select which version of the RobotWare you want to use. Only RobotWare versions that are valid for the used key are available.
4. Click **Next**.

Continues on next page

Entering the drive keys

To enter the keys for the drive modules:

1. In the **Drive Key** box, enter the key for the drive module. You can also click the **Browse** button and browse to the key file. If you used a virtual controller key, a virtual drive key is already generated by the wizard.
2. Click the right arrow button next to the **Drive Key** box. The key now appears in the **Added drive key** list.

   For real systems the drive key determines the connected robot model. For virtual systems you select the robot model in the *Modify Options* page. The default model is IRB140.
3. If you have a MultiMove system, repeat steps 1 and 2 for each drive key to add.

   If you have a MultiMove system, make sure that the keys are numbered in the same way as their corresponding drive modules are connected to the controller module. Use the up and down arrows to rearrange the drive keys, if necessary.
4. Choose whether you want to create the system as it is now, or to continue with the wizard.

   If you want to create the system as it is now, click **Finish**.

   If you want to modify options, or add options, parameter data or additional files to the home directory, click **Next**.

Adding additional options

Here you can add options, such as external axes and dispense applications, that are not included in the basic system. Options require a license key and must be first imported to the media pool. To add additional options, follow these steps:

1. In the **Key** box, enter the option key. You can also click the **Browse** button and browse to the option's key file.
2. Click the **Arrow** button.

   The option that the key unlocks is now displayed in the **Added Options** list.
3. Repeat steps 1 and 2 for all options you want to include.
4. Choose whether you want to create the system as it is now, or to continue with the wizard.

   If you want to create the system as it is now, click **Finish**.

   If you want to modify options, or add parameter data or additional files to the home directory, click **Next**.

Modifying options

Here you can set up and configure the options in your system. For virtual systems, you also select the robot models to use. To modify any options, follow these steps:

1. In the **Option** tree, expand the option folders to the level where you find the option you want to modify.

   Only the options unlocked by the used keys are available.
2. Modify the option.

Continues on next page

3. Repeat steps 1 and 2 for all options you want to modify.
4. Choose whether you want to create the system as it is now, or to continue with the wizard.

   If you want to create the system as it is now, click **Finish**.

   If you want to add parameter data or additional files to the home directory, click **Next**.

Adding parameter data

Parameter data is stored in the parameter data files (.cfg files). Each parameter topic has its own parameter file. You can add only one parameter file for each topic. To add parameter data, follow these steps:

1. In the **Parameter data** box, enter the path to the folder for the parameter data files. You can also click the **Browse** button and browse to the folder.
2. In the list of parameter data files, select the file you want to include and press the **Arrow** button. Repeat for all files you want to include.

   The included parameter data files will now appear in the **Added parameter data files** list.

   Repeat steps 1 and 2 for each parameter data file you want to add.
3. Choose whether you want to create the system as it is now, or to continue with the wizard.

   If you want to create the system as it is now, click **Finish**.

   If you want to add additional files to the home directory, click **Next**.

Adding files to the home directory

You can add any type of file to the system’s home directory. When the system is loaded to a controller, these files will also be loaded. To add files to the system's home directory, follow these steps:

1. In the **Files** box, enter the path to the folder for the files you want to include. You can also click the **Browse** button and browse to the folder.
2. In the list of files, select the file to add and click the **Arrow** button. Repeat for all files you want to add.

   The added files will now appear in the **Added files** list.
3. Choose whether you want to create the system as it is now, or to continue with the wizard.

   If you want to create the system as it is now, click **Finish**.

   If you want to read a summary before you create the system, click **Next**.

Completing the New Controller System Wizard

To complete the wizard, follow these steps:

1. Read the system summary.
2. If the system is OK, click **Finish**.

   If the system is not OK, click **Back** and make modifications or corrections.

6.4.4. Modifying a system

Overview

The **Modify Controller System Wizard**, used to modify existing systems, is launched from the System Builder.The wizard helps you with tasks like changing robots, adding and removing external axes and other options. A system that is running must be first shut down before modification.

Starting the wizard

To start the wizard when creating a new station:

1. If the system is currently running, on the **Controller** menu, point to **Shutdown** and then click **Shutdown**.
2. On the **Controller** menu, click **System Builder** to bring up a dialog box.
3. In the **System directory** list, enter or browse to the system directory. Select a system from the list beneath, review the system properties and add and save any comments.
4. In the **Actions** group, click **Modify**. This starts the wizard.
5. Read the information on the welcome page and click **Next**.

Modifying the program revision

The RobotWare versions that are available for the system are determined by the controller key. The key is essential to the system and cannot be modified.

To use another RobotWare version than the available ones, create a new system with another key.

To optionally modify the program revision, follow the appropriate step or steps:

1. To keep the current RobotWare version, select **Yes** and then click **Next** .
2. To replace the current RobotWare version, Select **No, replace it**..
3. In the **Media pool** box, enter the path to the media pool. You can also click the **Browse** button and browse to the folder.
4. In the **New program revision** box, select which version of RobotWare you want to use. Only RobotWare versions that are valid for the RobotWare key are available.
5. Click **Next**.

Continues on next page

Adding or removing drive keys

The drive key corresponds to the drive modules in your controller. For MultiMove systems, you have one drive module (and one key) for each robot. The keys for your system are delivered together with the controller.

the system is created with a vritual controller key, virtual drive keys are generated by the wizard. when you have added one virtual drive key for each robot, you select which robot to use for each key on the *Modify Options* page.

To optionally add or remove the keys for the drive modules, follow these steps:

1. To add a key for a drive module, enter the key in the **Enter Drive Key** box. You can also click the **Browse** button and browse to the key file.
2. Click the right arrow button. The key now appears in the **Added drive key** list.

   If you have a MultiMove system, repeat steps 1 and 2 for each drive key to add.
3. To remove a drive module, select the corresponding key in the **Added drive key** list and click **Remove drive key**.

   If you have a MultiMove system, repeat step 3 for each drive key to remove.
4. If you have a MultiMove system, make sure that the keys are numbered in the same way as their corresponding drive modules are connected to the controller module. Use the up and down arrows to rearrange the drive keys, if necessary.
5. Choose whether you want to create the system as it is now, or to continue with the wizard.

   If you want to create the system as it is now, click **Finish**.

   If you want to modify options, parameter data or add files to or remove files from the home directory, click **Next**.

Adding or removing additional options

To optionally add or remove additional options:

1. To add an addtional option, in the **Enter Key** box, enter the option key. You can also click the **Browse** button and browse to the option's key file.
2. Click the **Arrow** button.

   The option that the key unlocks is now displayed in the **Added Options** list.
3. Repeat steps 1 and 2 for all options you want to include.
4. To remove an additional option, iIn the **Added options** list, select the option you want to remove.
5. Click **Remove**.
6. Choose whether you want to create the system as it is now, or to continue with the wizard.

   If you want to create the system as it is now, click **Finish**.

   If you want to modify parameter data or add files to or remove files from the home directory, click **Next**.

Continues on next page

Modifying options

To optionally modify any options, follow these steps:

1. In the **Option** tree, expand the option folders to the level where you find the option you want to modify.

   Only the options unlocked by the used keys are available.
2. Modify the option.
3. Repeat steps 1 and 2 for all options you want to modify.
4. Choose whether you want to create the system as it is now, or to continue with the wizard.

   If you want to create the system as it is now, click **Finish**.

   If you want to modify parameter data or add files to or remove files from the home directory, click **Next**.

Adding or removing parameter data

Parameter data is stored in the parameter data files (.cfg files). each parameter topic has its own parameter file. You can add only one parameter file for each topic. To addor remove parameter data, follow these steps:

1. To add parameter data, in the **Parameter data** box, enter the path to the folder for the parameter data files. You can also click the **Browse** button and browse to the folder.
2. In the list of parameter data files, select the file you want to include and press the **Arrow** button. Repeat for all files you want to include.

   The included parameter data files will now appear in the **Added parameter data files** list.

   Repeat steps 1 and 2 for each parameter data file you want to add.
3. To remove parameter data, in the **Added parameter data files** list, select the parameter data file to remove.
4. Click **Remove**.
5. Choose whether you want to create the system as it is now, or to continue with the wizard.

   If you want to create the system as it is now, click **Finish**.

   If you want to add to or remove files from the home directory, click **Next**.

Add files to or remove files from the home directory

You can add any type of file to the system's home directory, or remove files from it. When the system is loaded to a controller, these files will also be loaded. To optionally add files to or remove files from the system's home directory, follow these steps:

1. To add files, in the **Files** box, enter the path to the folder for the files you want to include. You can also click the **Browse** button and browse to the folder.
2. In the list of files, select the file to add and click the **Arrow** button. Repeat for all files you want to add.

   The added files will now appear in the **Added files** list.
3. To remove files, in the **Added files** list, select the file to remove.

Continues on next page

4. Click **Remove**.
5. Choose whether you want to create the system as it is now, or to continue with the wizard.

   If you want to create the system as it is now, click **Finish**.

   If you want to read a summary before you create the system, click **Next**.

Complete the Modify Controller System wizard

To complete the wizard, follow these steps:

1. Read the system summary.
2. If the system is OK, click **Finish**.

   If the system is not OK, click **Back** and make modifications or corrections.

Result

Modifications will take effect when the wizard is completed.

If the system has been downloaded to a controller, it must be downloaded again before the modifications will take effect on the controller.

Deleting a system

To delete a system, follow this steps:

1. From the **System Builder** dialog box, select the system and then click **Delete**.

6.4.5. Copying a system

Copy a system

To copy a system, follow these steps:

1. From the **System Builder** dialog box, select the system and then click **Copy** to bring up a dialog box.
2. Enter a name for the new system and a path, and then click **OK**.

6.4.6. Creating a system from backup

Overview

The **Create System from Backup Wizard**, which creates a new system from a controller system backup, is launched form the System Builder. In addition, you can change the program revision and options.

Starting the wizard

To start the wizard, follow these steps:

1. From the **System Builder** dialog box, click **Create from Backup**. This starts the wizard.
2. Read the information on the welcome page and click **Next**.

Specifying the name and location

To specify the destination folder, follow these steps:

1. In the **Name** box, enter a name for the system you are creating.
2. In the **Path** box, enter the path to the system directory in which you will store the system.

   You can also click the **Browse** button and browse to the system directory.
3. Click **Next**.

Locating the backup

To locate a system from backup, follow these steps:

1. In the **Backup folder** box, enter the path to the backup folder. Alternatively, click the **Browse** button to browse to it. Click **Next**.
2. In the **Media Pool** box, enter the path to the media pool containing the appropriate RobotWare program. Confirm the backup information that now appears in the wizard. Click **Next**.

6.4.7. Downloading a system to a controller

Overview

All systems you access from the System Builder are stored on your computer. If you wish to run a system on a robot controller, you must first load it to the controller, which then requires a restart.

Load a system

To load a system to a controller, follow these steps:

1. From the System Builder dialog box, select a system and then click **Download to Controller** to bring up a dialog box.
2. Specify the Destination Controller for the system.

You can select by using the...	if...
Select controller from list option	the controller has been detected automatically.
Specify IP address or controller name option	your PC and the robot is connected to the same network. You can only use the controller name in DHCP networks.
Use service port option	your PC is directly connected to the controller's service port.

3. Optionally, click **Test Connection** to confirm that the connection between the computer and the Controller is OK.
4. Click **Load**.
5. Answer **Yes** to the question **Do you want to load the system**.

Yes	There will be a restart and the system is downloaded to controller.
No	There will be no restart. With the next restart the system will be downloaded to controller.
Cancel	The downloading is canceled.

6.4.8. Creating boot media

Overview

Boot media is an entire system which the System Builder packs to a single file and commonly stores on a hard disk or USB memory. The controller then accesses the file through its Ethernet port or USB port, respectively.

Creating boot media

To create boot media, follow these steps:

1. From the **System Builder** dialog box, select a system and then click **Boot Media**.
2. In the **Path** box, enter the path to the folder where you want to store the boot media file. Alternatively, browse to the location.
3. Click **OK**.

Result

To load the boot media system to a controller, first connect it and then restart the controller with the advanced restart method X-start.

6.4.9. Examples using the System Builder Offline

6.4.9.1. A MultiMove system with two coordinated robots

Overview

In this example we will use the System Builder to create a coordinated offline system with one IRB2400 and one IRB1600 robot to use in a new RobotStudio station.

Starting the New Controller System Wizard

To create a system like the one described above, follow these steps:

1. Click **System Builder** to bring up the dialog box.
2. In the dialog box, click **Create New** to bring up the **New Controller System Wizard**.
3. Read the welcome text, and click **Next** to continue to the next page.

Entering the name and path

1. In the **Name** box, enter the name of the system. The name must not contain blank spaces or non-ASCII characters.

   In this example, name the system *MyMultiMove*.
2. In the **Path** box, enter the path for the folder to save the system in, or click the **Browse** button to browse to the folder or create a new one.

   In this example, save the system in *C:\Program Files\ABB Industrial IT\Robotics IT\RobotStudio\ABB Library\Training Systems*.
3. Click **Next** to continue to the next page.

Entering the controller key

1. Select the **Virtual key** check box. A virtual controller key now appears in the Controller Key box. In this example we will use the default media pool and RobotWare version.
2. Click **Next** to continue to the next page.

Entering drive keys

1. Click the **Right Arrow** button next to the **Enter Drive key** box twice to create one drive key for each robot.
2. Click **Next** to continue to the next page.

Continues on next page

Adding options

This system does not require any additional option keys. Click **Next** and continue to the next page of the wizard.

Modifying options

When creating robot systems from real robot keys, the key sets the options. But since we are using a virtual key, we have to set the options manually.

To set the options necessary for a MultiMove, follow these steps:

1. Scroll down to the **RobotWare / Motion Coordination 1** group and select the **MultiMove Coordinated** check box.
2. Scroll down to the **RobotWare / I/O control** group and select the **Multitasking** and the **Advanced RAPID** check boxes.
3. Scroll down to the **DriveModule1 / Drive module application** group and expand the **ABB Standard manipulator** option. Select the **IRB 2400 Type A** option, manipulator variant **IRB 2400L Type A**.
4. Scroll down to the **DriveModule2 / Drive module application** group and expand the **ABB Standard manipulator** option. Select the **IRB 1600** option, manipulator variant **IRB 1600-5/1.2**.
5. Click **Finish** and the system will be created.

6.4.9.2. A system with support for one robot and one positioner external axis

Overview

In this example we will use the System Builder to create an offline system to use in a new RobotStudio station with one IRB1600 robot and one IRBP 250D positioner external axis.

Prerequisites

When creating systems for positioner external axes, you need the media pool and the license key file for that specific positioner. In this example we will use a media pool and license key file for a demo positioner.

Paths to files and folders assume that RobotStudio and the RobotWare media pool have been installed at their default locations on Windows XP. If not, adjust the paths accordingly.

Starting the New Controller System Wizard

To create a system like the one described above, follow these steps:

1. Click **System Builder** to bring up a dialog box.
2. In the dialog box, click **Create New** to bring up the **New Controller System Wizard**.
3. Read the welcome text, and click **Next** to continue to the next page.

Entering the controller key

1. Select the **Virtual key** check box. A virtual controller key now appears in the Controller Key box. In this example we will use the default media pool and RobotWare version.
2. Click **Next** to continue to the next page.

Entering drive keys

1. Click the **Right Arrow** button next to the **Enter Drive key** box to create one drive key for the robot.
2. Click **Next** to continue to the next page.

Adding options

This is where we point out the key file for the positioner.

1. Next to the **Enter key** box, click the browse button and select the key file.

   In this example, browse to and select the file *extkey.kxt* in the folder *C:\Program Files\ABB Industrial IT\Robotics IT\MediaPool\3HEA-000-00022.01*.

   TIP!

   In the *MediaPool* folder media pools for several standard positioners are installed. They are named by the positioner's article number, with a suffix that indicates if it is configured for single-robot or MultiMove systems.
2. Click the **Right Arrow** button next to the **Enter key** box to add the key for the positioner.
3. Click **Next** and continue to the next page of the wizard.

Continues on next page

Modifying options

When creating robot systems from real robot keys, the key sets the options. But since we are using a virtual key, we have to set the options manually. To set the options necessary for a positioner, follow these steps:

1. Scroll down to the **RobotWare / Hardware** group and select the **709-x DeviceNet** check box.

   This option is for the communication between the controller and the track external axis.
2. Scroll down to the **RobotWare / Motion coordination part 2** group and select the **Multiple Axis Positioner** check box.

   This option applies since the positioner has multiple axes.
3. Scroll down to the **RobotWare / I/O control** group and select the **Logical Cross Connection** check box.

   This option is for the communication between the virtual controller and the track external axis.
4. Scroll down to the **DriveModule1 / Drive module application** group and expand the **ABB Standard manipulator** option. Select the **IRB 1600** option.

   This option sets the robot to an IRB 1600-5/1.2.
5. Scroll down to the **DriveModule1 / Additional axes configuration** group and expand the **Add axes IRB/drive module 1600/2400/260** option and select the **R2C2 Add drive** option.

   Expand the **Drive type in position Z4** and select the**753-1 drive C in pos Z4** option.

   Expand the **Drive type in position Y4** and select the**754-1 drive C in pos Y4** option.

   Expand the **Drive type in position X4** and select the**755-1 drive C in pos X4** option.

   This option adds drive modules for the positioner axes.
6. Click **Finish** and the system will be created. When starting the system in a RobotStudio station, you have to set up the system to load a model for the positioner and to get the motions to work properly. See *Placing external axes on page 79* for more information.

6.4.9.3. Options settings for systems with positioners

Overview

This is an overview of the RobotWare options to set when creating a system for positioner external axes. Note that besides setting the RobotWare options, you must add an additonal option key for the positioner.

Media pools and option keys for the positioners

If you have the media pool and option key for your positioner, you can use these files.

If not, media pools for standard positioners are installed with RobotStudio. The path to these media pools in a default installation is: *C:\program files\ABB Industrial IT\Robotics IT\MediaPool*. In this folder a media pool for each positioner is located. These are named by the article number of the positioner, with a suffix that indicates if it is configured for a single-robot or a MultiMove system.

In the **Add additional options** page of the **System Builder**, you should add the option for the positioner by opening the mediapool folder for the positioner to add and selecting the *extkey.kxt* file.

Options for positioners in singe-robot systems

When adding a positioner to a single-robot system, the positioner will be added to the same task as the robot. Below, the options to set on the **Modify Options** page of the **System Builder** for such a system are listed:

- **RobotWare** > **Hardware** > **709-x DeviceNet** > **709-1 Master/Slave Single**
- **RobotWare** > **Motion coordination part 2** > **Multiple Axis Positioner**
- **RobotWare** > **I/O control** > **Logical Cross Connections**
- Optionally, for using the system with ArcWare also add **RobotWare** > **Application Arc** > **633-1 Arc**
- **DriveModule 1** > **Additional axes configuration** > **Add axes IRB/Drive module** for your robot model > **RC2C Add drive** > **753-1 Drive C in pos Z4** > **754-2 Drive T in pos Y4** > **755-3 Drive U in pos X4**

Continues on next page

Options for positioners in MultiMove robot systems

When adding a positioner to a MultiMove robot system, the positioner shall be added to a task of its own (thus you also have to add a drive key for the positioner). Below, the options to set on the **Modify Options** page of the **System Builder** for such a system are listed:

- **RobotWare** > **Hardware** > **709-x DeviceNet** > **709-1 Master/Slave Single**
- **RobotWare** > **Motion coordinated part 1** > **604-1 MultiMove Coordinated**
  Optionally, expand the MultiMove Coordinated option and select process options for the robots.
- **RobotWare** > **Motion coordination part 2** > **Multiple Axis Positioner**
- **RobotWare** > **I/O control** > **Logical Cross Connections**
- Optionally, for using the system with ArcWare, add **RobotWare** > **Application Arc** > **633-1 Arc**
- **DriveModule 1** > **Additional axes configuration** > **Add axes IRB/Drive module** for your robot model > **RC2C Add drive** > **753-1 Drive C in pos Z4** > **754-2 Drive T in pos Y4** > **755-3 Drive U in pos X4**. For the other drive modules, no additional axes should be configured.

6.5. Handle I/O

Overview

The I/O system handles input and output signals to and from the controller. Below are the parts of the system described, as well as common types of signals.

The I/O system window is used to view and set previously configured signals, and to activate and deactivate I/O units.

The I/O system

The I/O system of a controller consists of I/O buses, I/O units and I/O signals. The I/O buses are the controller's connections for I/O units (for instance I/O boards) and the I/O units contain channels for the actual signals.

The I/O buses and units are displayed in the robot view, as child nodes under each controller and the I/O signals are displayed in the I/O window.

I/O signals

I/O signals are used to communicate between the controller and external equipment, or to change variables within a robot program.

Input signals

Input signals notify something for the controller, for instance a feeder belt can set an input signal when it has positioned a work piece. The input signal can then be programmed to start a specific part of the robot program.

Output signals

The controller uses output signals to notify that a specified condition has been fulfilled. For instance, after the robot has finished its sequence, an output signal can be set. This signal can then be programmed to start a feeder belt, update a counter or trigger any other action.

Simulated signals

A simulated signal is a signal that is manually given a specific value that overrides the actual signal. Thus simulated signals might be useful for testing robot programs without activating or running equipment.

Virtual signals

Virtual signals are signals that are not configured to belong to a physical I/O unit. Instead, they reside inside the controller's memory. A common use for virtual signals is to set variables and store changes in a robot program.

Procedures

For using the I/O system window, see *Inputs / Outputs on page 282*.

For adding a signal, see *Add signals on page 292*.

6.6. Configure systems

Configuring system parameters

System parameters may be configured as follows:

- To view topics, types, instances and parameters
- To edit the parameters of an instance
- To copy and paste instances
- To add and delete instances
- To load and save complete configuration files to and from controllers

When working with configurations, the following tools, see *Configuration editor on page 297*, are useful:

Tool	Use
The Configuration Editor	With the Configuration Editor you work with the types and instances of a specific topic.
The Instance Editor	With the Instance Editor, you specify the values of the parameters in the instances of system parameter types.

NOTE!

To edit system parameters, you must have write access to the controller.

Terms

System parameters	The sum of all parameters that configure the system, these are divided into topics and types.
Topic	A collection of parameters relating to a specific area, and the highest level in the system parameter structure. Examples are Controller, Communication and Motion.
Type	A set of parameters for a specific configuration task. A type can be seen as a pattern describing the construction and properties for the parameters included in the task. For instance, the type *Motion System* defines which parameters shall be used for configuring a motion system.
Instance	An actualization of a type, an instance is a specific set of parameters with unique values created from a type pattern. In the Configuration Editor, each row in the Instance list is an instance of the type selected in the Type list.
Parameter	A property to set when configuring the robot system.
Configuration file	Contains all public parameters of a specific topic.

Viewing configurations

1. To view the topics of a controller, from the **Offline/Online** tab, expand the **Configuration** node for the controller.

   All topics in are now displayed as child nodes to the Configuration node.

2. To view the types and instances of a topic, double-click the topic node for the topic to view.

Continues on next page

The Configuration Editor is now opened, listing all types of the topic in the **Type name** list. In the **Instance** list, each instance of the type selected in the Type name list is displayed as row. The parameter values of the instances are displayed in the columns of the instance list.

3. To view detailed parameter information for an instance, double-click the instance.

   The instance editor now displays the current value, restrictions and limits of each parameter in the instance.

Editing parameters

You can either edit the parameters of one single instance, or you can edit several instances at one time. Editing several instances at one time is useful when you want to change the same parameter in several instances, like when moving signals from one I/O unit to another.

1. In the **Offline/Online** tab, expand the **Controller** and the **Configuration** node and double-click the topic that contains the parameters to edit.

   This opens the Configuration Editor.
2. In the **Type name** list of the Configuration Editor, select the type that the parameter to edit belongs to.

   The instances of the type is now displayed in the Instance list of the Configuration Editor.
3. In the **Instance** list, select the instances to edit and press the Enter Key. To select several instances at once, hold down the SHIFT or CTRL key while selecting.

   The Instance Editor is now displayed.
4. In the Parameter list of the Instance Editor, select the parameter to edit and change the value of the parameter in the **Value** box.

   When editing several instances at one time, the parameter values you specify will be applied to all instances. For parameters that you do not specify any new value, each instance will keep its existing value for that parameter.
5. Click **OK** to apply the changes to the configuration database of the controller.

   For many parameters, the changes will not take affect until the controller is restarted. If your changes require a restart, you will be notified of this.

   You have now updated the controller's system parameters. If the changes require a restart of the controller, the changes will not take affect until you do this. If you are going to make several changes, you can wait with the restart until all changes are done.

Adding instances

With the Configuration Editor, you can select a type and create a new instance of it. For example, adding a new instance of the type Signal creates a new signal in the system.

1. In the **Offline/Online** tab, expand the **Controller** and the **Configuration** node and double-click the topic that contains the type of which you want to add an instance.

   This opens the Configuration Editor.
2. In the **Type name** list of the Configuration Editor, select the type of which you want to add an instance.

Continues on next page

3. On the **Controller** menu, point to **Configuration** and click **Add type** (the word type is replaced by the type you selected previously).

   You can also right-click anywhere in the configuration editor and then select **Add type** from the shortcut menu.

   A new instance with default values is added and displayed in the **Instance Editor** window.
4. If required, edit the values.
5. Click **OK** to save the new instance.

   The values in the new instance are now validated. If the values are valid, the instance is saved. Otherwise, you will be notified of which parameter values to correct.

   For many instances, the changes will not take affect until the controller is restarted. If your changes require a restart you will be notified of this.

   You have now updated the controller's system parameters. If the changes require a restart of the controller, the changes will not take affect until you do this. If you are going to make several changes, you can wait with the restart until all changes are done.

Copying an instance

1. In the **Offline/Online** tab, expand the **Controller** and the **Configuration** node and double-click the topic that contains the instance to copy.

   This opens the Configuration Editor.
2. In the **Type name** list of the Configuration Editor, select the type of which you want to copy an instance.
3. In the **Instance** list, select one or several instances to copy.

   If you select several instances and they don't have the same value for all parameters, these parameters will have no default values in the new instances.
4. On the **Controller** menu, point to **Configuration** and click **Copy Type** (the word type is replaced by the type you selected previously).

   You can also right-click the instance to copy and then select **Copy Type** from the shortcut menu.

   A new instance with the same values as the one you copied is added and displayed in the **Instance Editor** window.
5. Change the name of the instance. If required, also edit the other values.
6. Click **OK** to save the new instance.

   The values in the new instance are now validated. If the values are valid, the instance is saved. Otherwise, you will be notified of which parameter values to correct.

   For many instances, the changes will not take affect until the controller is restarted. If your changes require a restart you will be notified of this.

   You have now updated the controller's system parameters. If the changes require a restart of the controller, the changes will not take affect until you do this. If you are going to make several changes, you can wait with the restart until all changes are done.

Continues on next page

Deleting an instance

1. In the **Offline/Online** tab, expand the **Controller** and the **Configuration** node and double-click the topic that contains the type of which you want to delete an instance.

   This opens the Configuration Editor.
2. In the **Type name** list of the Configuration Editor, select the type of which you want to delete an instance.
3. In the **Instance** list, select the instance to delete.
4. On the **Controller** menu, point to **Configuration** and then click **Delete type** (the word type is replaced by the type you selected previously).

   You can also right-click the instance to delete and then select **Delete type** from the shortcut menu.
5. A message box is displayed, asking if you want to delete or keep the instance. Click **Yes** to confirm that you want to delete it.

   For many instances, the changes will not take affect until the controller is restarted. If your changes require a restart you will be notified of this.

   You have now updated the controller's system parameters. If the changes require a restart of the controller, the changes will not take affect until you do this. If you are going to make several changes, you can wait with the restart until all changes are done.

Save one configuration file

The system parameters of a configuration topic can be saved to a configuration file, stored on the PC or any of its network drives.

The configuration files can then be loaded into a controller. They are thereby useful as backups, or for transferring configurations from one controller to another.

1. In the **Offline/Online** tab, expand the **Configuration** node and select the topic to save to a file.
2. On the **Controller** menu, point to **Configuration** and select **Save System Parameters**.

   You can also right-click the topic and then select **Save System Parameters** from the shortcut menu.
3. In the **Save As** dialog box, browse for the folder to save the file in.
4. Click **Save**.

Saving several configuration files

1. In the **Offline/Online** tab, select the **Configuration** node.
2. On the **Controller** menu, point to **Configuration** and click **Save System Parameters**.

   You can also right-click the configuration node and then click **Save System Parameters**.
3. In the **Save System Parameters** dialog box, select the topics to save to files. Then click **Save**.
4. In the **Browse for Folder** dialog box, browse for the folder to save the files in, and then click **OK**.

   The selected topics will now be saved as configuration files with default names in the specified folder.

Continues on next page

Loading a configuration file

A configuration file contains the system parameters of a configuration topic. They are thereby useful as backups, or for transferring configurations from one controller to another.

When loading a configuration file to a controller, it must be of the same major version as the controller. For instance, you cannot load configuration files from an S4 system to an IRC 5 controller.

1. In the **Offline/Online** tab, select the **Configuration** node.
2. On the **Controller** menu, point to **Configuration** and select **Load Parameters**.

   You can also right-click the configuration node and then select **Load Parameters** from the context menu.

   This opens the Select mode dialog box.
3. In the Select mode dialog box, select how you want to combine the parameters in the configuration file to load with the existing parameters:

If you want to	then
replace the entire configuration of the topic with the one in the configuration file.	select **Delete existing parameters before loading**
add new parameters from the configuration file to the topic, without modifying the existing ones.	click **Load parameters if no duplicates**
add new parameters from the configuration file to the topic and update the existing ones with values from the configuration file. Parameters that only exist in the controller and not in the configuration file will not be changed at all.	click **Load parameters and replace duplicates**

4. Click **Open** and browse to the configuration file to load. Then click **Open** again.
5. In the information box, click **OK** to confirm that you want to load the parameters from the configuration file.
6. When the loading of the configuration file is finished, close the Select mode dialog box.

   If a restart of the controller is necessary for the new parameters to take affect, you will be notified of this.

6.7. Handle events

Overview

An event is a message that notifies you that something has happened to the robot system, be it merely a change in operation mode or a severe error that calls for your immediate attention. If the event requires any action from you, this is stated in the event.

Events are displayed in the event logs of the FlexPendant and RobotStudio.

The event log keeps you informed of system status, allowing you to:

- view controller events.
- filter events.
- sort events.
- get detailed information about an event.
- save event log files on your PC.
- clear event records.

Event Log list

The event log list consists of all events matching your filter settings, with the following information for each event:

Type	The event type is an indication of the severity of the event.
Code	The event code is a number that identifies the event message.
Title	The event title is a short event description.
Category	The event category is an indication of the source of the event.
Seq. Number	The sequential number indicates the chronological order of the event.
Date and Time	Date and time is when the event occurred.

When you select an event in the list, detailed information will appear to the right.

Event type

The event type is an indication of the severity of the event.

There are three types of events:

Event type	Description
Information	A normal system event, such as starting and stopping programs, change of operating mode, motors on/off and so on. Information messages never require any action from you, but can be useful for error tracking, statistics collecting or monitoring user triggered event routines.
Warning	An event that you need to be aware of, but not so severe that the process or RAPID program needs to be stopped. Warnings, however, often indicate underlying problems that sooner or later must be solved. Warnings must sometimes be acknowledged.
Error	An event that prevents the robot system from proceeding. The running process or RAPID program cannot continue, but is stopped. All errors must be acknowledged. Most errors also require some immediate action from you in order to solve the problem.

Continues on next page

NOTE!

This information is also indicated by color: blue for information, yellow for warning and red for an error which needs to be corrected in order to proceed

Event code

The event code is a number that identifies the event message. Together with the event date and time each event has a unique identity.

Event title

The event title is a short description of the event.

Event category

The category is an indication of the source of the event.

Category	Display
Common	All recent events.
Operational	Events related to changes in operation or operating mode.
System	Events related to the current system.
Hardware	Events related to controller hardware.
Program	Events related to the running process applications and RAPID programs.
Motion	Events related to the movement of robots or other mechanical units .
I/O & Communication	Events related to input and output signals, serial or network communication and process buses.
User	Custom messages that have been programed into RAPID programs.
Internal	Internal low-level controller errors for ABB service personnel.
Process	Events related to Industrial Processes options., such as Spot, Arc and Dispense.
Cfg	Errors in a configuration file.

Depending on how the system is configured, additional categories may exist.

Sequential number

The sequential number indicates the chronological order of the event; the higher the number the more recently the event occurred.

Date and time

Date and time indicate exactly when the event occurred. Along with the event code, this timestamp guarantees that each event has a unique identity.

Event description

When you select an event in the list, detailed information about it will be displayed to the right. This includes a description, and when appropritate also consequences, causes and suggested actions to solve the problem.

Continues on next page

Overview

The Event Log automatically logs all controller events once it is started. By default, events are displayed in the chronological order specified by **Seq Numbers**.

NOTE!

Any modifications to the list you see will never affect the event log of the controller. What you see is just a copy.

Managing events

1. In the Robot View Explorer, select a system.
2. Double-click the **Events** node.

To sort events	Click the header for the column you want to sort by. To switch between ascending and descending sorting, click the header once again.
To filter events	In the **Category** list select the event category you want displayed.
To clear the event log	Click **Clear**. This will not affect the event log of the robot controller. It might still be impossible, however, to retrieve all events from a cleared record once again, as the oldest ones may have been erased from the controller hard disk due to lack of space. It is therefore recommended to save the record to a log file before clearing.
To save all events to a single log file on the computer	Check the **Log to file** check box. If it remains checked, the log file will be updated with new events as they occur.
To save events of one or several categories to files on the computer	Click **Save** and then make your category choice. Specify the location for the log file(s) in the **Browse For Folder** dialog and then click **OK**. If you select **All** when selecting categories, a log file for each event category will be created.

Retrieving controller events

To clear the list and retrieve all existing events from the robot controller:

1. Optionally, save the existing Event Log record.
2. Select whether you want the list to be updated when new events occur , or if you are only interested in viewing events that have already occurred.

To ...	...then...
get automatic updates when new events occur	check the **Auto Update** check box. (Checked by default.)
say no to automatic updates when new events occur	clear the **Auto Update** check box.

3. Click **Get** to clear the current list, fetch and display all events that are currently stored in the controller log files.

6.8. Handle devices

Overview

The Device Browser displays the properties and trends of the various hardware and software devices in a robot controller. It is launched from the **Properties** command on the **Online** tab.

For procedures, see *Device Browser on page 316*.

6.9. RAPID Watch Online

Overview

The RAPID Watch Online window displays the variable status during the program execution. It has the following tabs:

Tab	Description
Name	Displays variable name
Value	Displays variable value
Type	Displays type of datatype
Source	Displays system name

NOTE!

When running a program in continuos execution mode, the contents of RAPID Watch Online window does not update until program execution stops.

7 The Application Menu

7.1. Overview

Overview

The Application menu is accessed from the RobotStudio button in the upper left corner of the GUI.

7.2. New Station

Creating a station with a template system

1. Click **New Station** to bring up a dialog box.

   The **Template System** icon is already selected by default.
2. In the **Select Template System** list, either select an appropriate template or click **Browse** and browse to one.
3. In the **System** group, enter a name and location, and then click **OK**.

Creating a station with an existing system

1. Click **New Station** to bring up a dialog box.
2. Click the **Existing system** icon to bring up the **Existing System** dialog box.
3. In the **Select System Pool** list, select a folder.
4. In the **Systems Found** list, select a system.
5. Click **OK**.

Creating an empty station

1. Click **New Station** to bring up a dialog box.
2. Click the **Empty Station** icon. The station will be created and visible in the graphics window.

7.3. Screenshot

Prerequisites

For optimal results, first configure the options, see *Options:General:Screenshot on page 179*.

Capturing the screen

1. Click **Screenshot**.

   If the image is saved to file, this is reported in the output window.

7.4. Pack & Go

Packing a station

1. Click **Pack & Go** to open the **Pack & Go Wizard**.
2. On the **Welcome to the Pack & Go Wizard** page, click **Next**.
3. On the **Destination** page, specifiy the destination directory of the package by either entering the path in the box or browsing to it. Click **Next**.
4. On the **Libraries** page, select one of the three options. Click **Next**.
5. On the **Systems** page, select the check box to include backups of all robot systems. Optionally, select the check box to include a media pool for additional options. Click **Next**.
6. On the **Ready to pack** page, review the information and then click **Finish**.
7. On the **Pack & Go succeeded** page, review the results and then click **Close**.

7.5. Unpack & Work

Unpacking a station

1. Click **Unpack & Work** to open the **Unpack & Work Wizard**.
2. On the **Welcome to the Unpack & Work Wizard** page, click **Next**.
3. On the **Select package** page, specifiy the package to be unpacked by either entering the path in the box or browsing to it. Specify as well the destination directory of the files. Click **Next**.
4. On the **Controller Systems** page, specify the path to the media pool and the RobotWare version. Optionally, select the check box to automatically restore backup. Click **Next**.
5. On the **Ready to unpack** page, review the information and then click **Finish**.
6. On the **Unpack & Work succeeded** page, review the results and then click **Close**.

7.6. Station Viewer

Overview

The Station Viewer can playback a station in 3D on computers that do not have RobotStudio installed. It packages the station file together with files needed to view the station in 3D. It can also play recorded simulations.

Prerequisites

- Works on Windows XP SP2 and Windows Vista
- .NET Framework 2.0 must be installed on the playback computer

Creating and loading a Station Viewer

1. To create a Station Viewer, on the **Application menu,** click **Collaborate.**
2. Select **Save Station as Viewer** to bring up a dialog box.
3. Specify a file name and save as **.exe** file.

   NOTE! Select the option **Show comments on startup** and add text in the box to view the comment when the Station Viewer is started.

   NOTE! Select the option **Include last executed simulation** to include a recording of the simulation. By default this option is disabled.
4. To load a Station Viewer, double-click the package (**.exe**) file on the target computer.

   The results are displayed in the Output window and the embedded station file is automatically loaded and presented in a 3D view.

   NOTE! You can open any station created in RobotStudio 5.12 with the Station Viewer. You can pan, rotate, and navigate in the graphical viewer.

Configuring user settings of a Station Viewer

To configure the user settings of a Station Viewer, on the **Application menu**, click **RobotStudio Options**.

Command Buttons

Apply	Click this button to save all options in the current page.
Reset	Click this button to reset to the settings you had before this session all values that you have changed on the current page.
Default	Click this button to reset to their default values all settings on the current page.

Options:General:Appearance

Select application language	Select the language to be used. **NOTE!** The default language is the same as that of the target user's operating system if available, otherwise it is English.
Select color theme	Select the color to be used.

Continues on next page

Options:General:Graphics

Select API to use for 3D graphics	Select the low-level API to be used for rendering 3D graphics.The default value is Direct3D, but OpenGL might be faster or stabler. **NOTE!** No need to restart when changing this option.
Background color	Select the color from the color theme, or from the color stored in the stations.

Simulation

When you run a simulation, the movements and visibility of objects are recorded. This recording is optionally included in the Station Viewer.

Simulation control buttons are enabled when the Station Viewer contains a recorded simulation.

Following are the Simulation control buttons:

Play	Starts or resumes simulation playback
Stop	Stops simulation playback
Reset	Resets all objects to their initial state and process time display to zero
Run mode	Select to run the simulation once or continuously
Process time	Displays the current simulation time

NOTE! If the simulation executes a VSTA macro, it is not reflected when playing back the simulation in the Station Viewer.

7.7. RobotStudio Options

Common buttons

Apply	Click this button to save all options in the current page.
Reset	Click this button to reset to the settings you had before this session all values that you have changed on the current page.
Default	Click this button to reset to their default values all settings on the current page.

Options:General:Appearance

Select application language	Select the language to be used.
Select color theme	Select the color to be used.
Display Position Edit boxes with Red/Green/ Blue background	Select the check box if you want to display the position boxes in the modify dialog boxes with colored background. Default value: selected.

Options:General:Licensing

View installed license keys	Click to view the license keys per feature, version, type, expiration date and status.
Request a license	Click to request a license.
Install a license	Click to install a license.

Options:General:Units

Quantity	Select the quantity for which you want to change the units.
Unit	Select the unit for the quantity.
Display decimals	Enter the number of decimals that you want to be displayed.
Edit decimals	Enter the number of decimals that you want when modifying.

Options:General:Advanced

Enable Windows Error Reporting	Select this check box to send error data directly to a Microsoft server to be downloaded and analyzed by ABB support. Available only on Windows XP or later.
Number of undo/redo steps	The number of operations that can be undone or redone. Lowering this value can decrease memory usage.
Show acknowledge dialog box when deleting objects	Warns when deleting objects.
Show acknowledge dialog box when deleting targets and correspond-ing move instructions	Warns when deleting targets and move instructions.
Warn about running Virtual Controller processes on startup	Warns of orphaned VC processes.

Continues on next page

Options:General:Files & Folders

User Project Folder	Enter the path to your project folder. This will be the folder displayed in the open and save dialog boxes in RobotStudio.
...	To browse for your project folder, click the browse button.
Automatically create document subfolders	Select this check box to enable the creation of individual subfolders for document types.
Enable Autosave	Select the check box to automatically save the station with defined intervals. Default value: cleared
Interval	Specify the interval between the savings when using Autosave in this box.

Options:General:Screenshot

Entire application window	Select this option to capture the entire application.
Active document window	Select this option to capture the active document window, typically the graphics window.
Copy to clipboard	Select this check box to save the captured image to the system clipboard.
Save to file	Select this check box to save the captured image to file.
Location	Specify the location of the image file. The default location is the "My Pictures" system folder.
...	Browse for the location.
File name	Specify the name of the image file. The default name is "Robot-Studio" to which is added a date.
The file suffix list	Select the desired file format. The default format is JPG.

Options:General:Screen Recorder

Framerate	Specify the framerate in frames per second.
Same as window	Select this option to use the same resolution as in the graphics window.
Limit resolution	Select this option to scale down the resolution.
Maximum width	Specify the maximum width in pixels.
Maximum height	Specify the maximum height in pixels.
Output file format	Select the output file format. The default format is AVI.
Video compression	Select the video compression format. **NOTE!** DivX format is not supported.
Start recording after	Select this option to start recording after the specified time.
Stop recording after	Select this option to stop recording after the specified time.

Options:Robotics:Editor

Wrap long lines	Select this check box if you want to wrap long lines.
Text styles	Specify the appearance of the various text classes.

Continues on next page

Options:Robotics:Rapid

Show dialog when warning for globally defined workobjects	Select this check box if you want RobotStudio to display a warning when there are workobjects with the same name that have been declared as in other tasks. Default value: selected.
Show synchronize dialog box after loading program /module	Select this check box if you want the synchronize dialog box to be displayed when you have loaded a program or a module. Default value: selected.
Show notification that default data is used	Select this check box if you want to be notified that *wobj0* and/ or *tool0* is active and will be used in the current action. Default value: selected.
Show synchronization notification	Default value: selected.
Set as active when creating tooldata	Select this check box if you want newly created tooldata to be set as active. Default value: selected.
Set as active when creating workobjects	Select this check box if you want newly created workobjects to be set as active. Default value: selected.

Options:Robotics:Synchronization

Use default synchronization locations	Converting data, such as target to Workobject, shall use the default behavior for synchronization locations. Default value: selected.
Show default synchronization locations notification	Notifies of the behavior above. Default value: selected.
Declaration default locations	Specify the locations for corresponding objects when synchronizing to the VC.

Options:Robotics:Mechanism

Approach Vector	Select the approach vector. Default value: Z.
Travel Vector	Select the travel vector. Default value: X.
Enable configuration check for jump to target/ move instruction	Select this check box if you want to enable the configuration check configurations when jumping to target or move instructions. When selected and a target does not have a validated configuration assigned, you will be asked to set one. When cleared, the configuration closest to the current one is used. Default value: selected.

Options:Robotics:Virtual FlexPendant

Always on top	Select this check box if you want to have the virtual FlexPendant always on top. Default value: selected.
Enable transparency	Select this check box if you want parts of the virtual FlexPendant to be transparent. Default value: selected.
Show virtual Operator Window	Select this check box to enable the Operator window feature. Default value: Disabled.

Options:Online:Authentication

Recent Users	Lists the recent users.

Continues on next page

Remove/Remove All	Click these buttons to remove one or all recent users, respectively.
Enable Automatic Logoff	Select the check box if you want to log off automatically.
Timeout	Determines the length of the session before being automatically logged off.

Options:Graphics:Renderer

Select API to use for 3D graphics	Select the low-level API to be used for rendering 3D graphics. The default value is **OpenGL**, but **Direct3D** might be faster or stabler.

Options:Graphics:Appearance

Background color	Click the colored rectangle to change the background color.
Gradient	Select the check box if you want to display the background color in gradient style. Default value: cleared.
Show floor	Select the check box if you want the floor (at z=0) to be displayed. Change the floor color by clicking the colored rectangle. Default values: selected.
Color	Click the colored rectangle to change the color of the floor.
Transparent	Select the check box if you want the floor to be transparent. Default values: selected.
Show UCS Grid	Select the check box if you want the UCS grid to be displayed. Default value: selected.
Grid Space X	Change the UCS grid space in the X coordinate direction by entering the requested value in the box. Default value: 1000 mm (or equivalent in other units).
Grid Space Y	Change the UCS grid space in the Y coordinate direction by entering the requested value in the box. Default value: 1000 mm (or equivalent in other units).
Show coordinate system	Select the check box if you want the coordinate systems to be displayed. Default value: selected.

Options:Graphics:Performance

Detail level	Select if the detail level is to be Auto, Fine, Medium or Coarse. Default value: Auto.
Cull back-facing triangles	Select the check box if you want to ignore the back-facing triangles. Default value: selected. Culling back-facing triangles improves the graphics performance but may give unexpected display if surfaces in models are not faced correctly.
Enable twosided lighting	Select the check box if you want to enable two-sided lighting. Default value: cleared.
Cull objects smaller than	Select the size in pixels under which objects will be disregarded. Default value: 2 pixels.
Store model data on graphics card (recommended)	Select the check box if you want to store the model data on a graphics card. Default value: cleared. Storing model data on a graphics card improves graphics performance, but causes stability problems for some hardware combinations.

Continues on next page

The settings you make here are generic for all objects in RobotStudio. With the **Graphic Appareance** dialog box you can, however, override some of these settings for single objects.

Options:Graphics:Behavior

Navigation sensitivity	Select the navigation sensitivity when using the mouse movements or navigation buttons by clicking the bar and dragging it into position. Default value: 1.
Selection radius (pixels)	Change the selection radius (that is, how close the mouse cursor click must be to an item to be selected) by entering the requested pixel value in the box. Default value: 5.
Selection highlight	Set if the selected object shall be distinguished in the **Graphics** window by a color, by an outline or not at all. Default value: color.
Highlight color	Click the colored rectangle to change the highlight color.
Activate selection preview	Select the check box to enable temporarily highlighting of items that may be selected when the mouse cursor passes over them. Default value: selected.
Show local coordinate system for selected objects	Select the check box to show the local coordinate system for the selected objects. Default value: selected.

Options:Graphics:Geometry

Normal Tolerance	Enter the maximum deviation of surface normals in the **Fine**, **Medium** or **Coarse** boxes. Default values (in deg): Fine: 10, Medium: 22.5, Coarse: 45 (or equivalent in other units).
Surface Tolerance	Enter the maximum spatial deviation of surfaces in the **Fine**, **Medium** or **Coarse** boxes. Default values (in deg): Fine: 2, Medium: 10, Coarse: 50 (or equivalent in other units).
Curve Tolerance	Enter the maximum spatial deviation of curves in the **Fine**, **Medium** or **Coarse** boxes. Default values (in deg): Fine: 0.2, Medium: 1, Coarse: 5 (or equivalent in other units).

Options:Simulation:Collision

Perform collision detection	Select if collision detection is to be performed during simulation or always. Default value: During simulation.
Stop simulation at collision	Select this check box if you want the simulation to stop at a collision. Default value: cleared.
Log collisions to Output window	Select this check box if you want the collisions to be logged to the output window. Default value: selected.
Log collisions to file:	Select this check box if you want to log the collisions to a file. Browse for the file to log in by clicking the browse button. Default value: cleared.
Enable fast collision detection	Select this check box to enhance the performance by detecting collisions between geometrical bounding boxes instead of geometrical triangles. This might result in falsely reported collisions, since the triangles are the true geometry and the bounding boxes always are larger. All true collisions will, however, be reported. The larger the object, the greater the number of false collisions that are likely to be detected.
View	Click this button to open the log file specified in the file box in Notepad.
Clear	Click this button to delete the log file specified in the file box.

Continues on next page

...	Click this button to browse for the file in which you want to log the collisions.

Options:Simulation:Virtual Time

Virtual Time mode- Free run	This option makes RobotStudio always use the free run mode.
Virtual Time mode - Time Slice	This option makes RobotStudio always use the time slice mode.

Options:Simulation:Accuracy

Simulation speed	Sets the simulation speed relative to real time.
Simulation timestep	Specifies the simulation timestep.

8 The Home Tab

8.1. Overview

The Home tab

The Home tab contains the controls required for building stations, creating systems, programming paths and placing items.

8.2. ABB Library

About this button

With this button, you can choose robots, positioners and tracks from their respective galleries.

8.3. Import Library

About this button

With this button, you can import equipment, geometries, positioners, robots, tools and training objects to your station libraries.

Click the lower split button to display galleries of tools, conveyors, cabinets and other equipment.

8.4. Robot System

About this button

With the **Robot System** button, you can either create a system from layout or template, choose an existing system, or select a system from a robot gallery and setup a conveyor tracking mechanism.

Creating a system from layout

1. Click **From Layout** to bring up the first page of the wizard.
2. In the **Name** box, enter the name of the system.
3. In the **Location** box, enter the path to the folder where the system will be stored. Alternatively, click **Browse** and browse to the folder.
4. In the **Media Pool** box, enter the path to the media pool. Alternatively, click **Browse** and browse to the folder.
5. In the **RobotWare Version** list, select the version of RobotWare you want to use.
6. Click **Next**.
7. In the **Mechanisms** box, select the mechanisms that you want to include in the system.
8. Click **Next**.

   The wizard now proposes a mapping of the mechanisms to a specific motion task, in accordance with the following rules:

   - Only one TCP robot is allowed per task.
   - Up to six motion tasks may be added, but only four TCP robots can be used, and they must be assigned to the first four tasks.
   - The number of tasks may not exceed the number of mechanisms.
   - If the system contains one TCP robot and one external axis, they will be assigned to the same task. It is, however, possible to add a new task and assign the external axis to it.
   - If the system contains more than one TCP robot, any external axes will be assigned to a separate task. It is, however, possible to move them to other tasks.
   - The number of external axes in a task is limited by the number of available drive modules in the cabinet (one for large robots, two for medium, three for small).

   Note! If only one mechanism was selected in the previous page, this page will not be shown.

   Tasks can be added and removed using the respective buttons; mechanisms can be moved up or down using the respective arrows. To map the mechanisms to tasks, follow this step:
9. Optionally, make any edits in the mapping, and then click **Next**.
10. Verify the summary and then click **Finish**.

    If the system contains more than one robot, the number of tasks and the baseframe positions of the mechanism should be verified in the System Configuration window.

Continues on next page

Adding a template system

1. Click **From Template** to bring up a dialog box.
2. In the **Select Template System** list, either select an appropriate template or click **Browse** and browse to one.
3. In the **Libraries** group, select whether to import libraries or to use the existing station libraries.
4. In the **System** group, enter a name and location, and then click **OK**.

Adding an existing system

1. Click **Existing** to bring up a dialog box.
2. In the **Select System Pool** list, select a folder.
3. In the **Systems Found** list, select a system.
4. In the **Libraries** group, select whether to import libraries or to use the existing station libraries.
5. Click **OK**.

Selecting a system from a robot gallery

1. Click **Quick System** to bring up a gallery, and then click the appropriate robot.

Setting up a conveyor

1. Click **Setup**.
2. In the **Part Sequence** tab, select **Part** from **Available Part**s.

   The right arrow button is enabled.
3. Click right arrow button to move the **Part** to **Parts moved by Conveyor** list.
4. Click up and down arrow buttons to move the selected part in **Parts moved by Conveyor** list.
5. In the **Part Tracking** tab, select **Part** from **Parts moved by Conveyor** list.
6. Select **CNV1** from the **Mechanical Unit** list.
7. Select a workobject from the **Workobject** list.
8. Click **Add.** The workobject appears in the list.

   NOTE! If the same workpiece is tracked by more than one robot, add a pair of workobject for each robot that tracks the workpiece. This procedure has to be repeated for each workpiece that should be tracked.
9. Click **OK**.
10. Activate the Conveyor Mechanical Unit (CNV1). See *Activate Mechanical Units on page 264*

Continues on next page

Removing objects from conveyor

1. Click **Setup**.

   The Conveyor Setup dialog box appears.

2. In the **Part Sequence** tab, select **Part** from the **Parts moved by Conveyor** list

   The left arrow button is anabled.

3. Click left arrow to remove the part from the **Parts moved by Conveyor** list to **Available Parts** list.

8.5. Import Geometry

Importing a geometry

1. Click **Import Geometry**.
2. Browse to the library where the geometry is located.

   For predefined geometries, click the **ABB Library** icon to the left in the dialog box.
3. Select the requested geometry and **Detail Levels**. Click **Open**.

   If you want the geometry to move with another object, attach it to the requested object, see *Attach to on page 323*.

   To modify the detail level for import of geometries, see *RobotStudio Options on page 178*.

8.6 Frame

8.6.1. Frame

Creating a frame

1. Click **Frame**.
2. In the dialog box, specify the positions for the frame.

Reference	Select the **Reference** coordinate system to which all positions or points will be related.
Frame Position	Click in one of these boxes, and then click the frame position in the graphics window to transfer the values to the **Frame Position** boxes.
Frame Orientation	Specify the coordinates for the frame orientation.
Set as UCS	Select this check box to set the created frame as the user coordinate system.

8.6.2. Frame from Three Points

Creating a frame from three points

1. Click **Frame from Three points** to bring up a dialog box.
2. Decide how you want to specify the frame:

To specify the frame using	Select
X, Y and Z coordinates, a point on the X axis and a point in the X-Y plane	**Position**
two points on the X axis and one point on the Y axis	**Three Point**

3. If you select **Position**:
   - Enter the **Position** for the object.
   - Enter the **Point on X axis** for the object.
   - Enter the **Point on X-Y plane** for the object.
   - Click **Create**.
4. If you select **Three Point**:
   - Enter the **First Point on X axis** for the object. This is the point closest to the frame's origin.
   - Enter the **Second Point on X axis** for the object. This is the point further away in the positive X direction.
   - Enter the **Point on Y axis** for the object.
   - Click **Create**.

The Create Frame From Three Points dialog box

Position	Select this option if you want to create the frame by using a position and two points.
Frame Position	Click in one of these boxes, and then click the frame position in the graphics window to transfer the values to the **Frame Position** boxes.
Point on X axis	Click in one of these boxes, and then click the point position in the graphics window to transfer the values to the **Point on X axis** boxes.
Point on X-Y plane	Click in one of these boxes, and then click the point position in the graphics window to transfer the values to the **Point on X-Y plane** boxes.
Three Point	Select this option if you want to create the frame by using three points.
First Point on X axis	Click in one of these boxes, and then click the point position in the graphics window to transfer the values to the **First Point on X axis** boxes.
Second Point on X axis	Click in one of these boxes, and then click the point position in the graphics window to transfer the values to the **Second Point on X axis** boxes.

Continues on next page

Point on Y axis	Click in one of these boxes, and then click the point position in the graphics window to transfer the values to the **Point on Y axis** boxes.
Set as UCS	Select this check box to set the created frame as the user coordinate system.

8.7. Workobject

Creating a workobject

1. Click **Workobject** to bring up a dialog box.
2. In the **Misc Data** group, enter the values for the new workobject.
3. In the **User Frame** group, do one of the following:
   - Set the position of the user frame by entering values for the **Position x, y, z** and the **Rotation rx, ry, rz** for the workobject by clicking in the **Values** box.
   - Select the user frame by using the **Frame by points** dialog box.
4. In the **Object Frame** group you can reposition the object frame relative to the user frame by doing any of the following:
   - Set the position of the object frame by selecting values for **Position x, y, z** by clicking in the **Values** box.
   - For the **Rotation rx, ry, rz**, select **RPY (Euler XYX)** or **Quaternion**, and enter the rotation values in the **Values** dialog box.
   - Select the object frame by using the **Frame by points** dialog box.
5. In the **Sync Properties** group, enter the values for the new workobject.
6. Click **Create**. The workobject will be created and displayed under the **Targets** node under the robot node in the **Paths&Targets** browser.

The Create Workobject dialog box

Name	Specify the name of the workobject.
Robot holds workobject	Select whether the workobject is to be held by the robot. If you select **True**, the robot will hold the workobject. The tool can then either be stationary or held by another robot.
Moved by mechanical unit	Select the mechanical unit that moves the workobject. This option is applicable only if **Programmed** is set to **False**.
Programmed	Select **True** if the workobject is to use a fixed coordinate system, and **False** if a movable (that is, external axes) will be used.
Position x, y, z	Click in one of these boxes, and then click the position in the graphics window to transfer the values to the **Position** boxes.
Rotation rx, ry, rz	Specify the rotation of the workobject in the UCS.
Frame by points	Specify the frame position of the user frame.
Position x, y, z	Click in one of these boxes, and then click the position in the graphics window to transfer the values to the **Position** boxes.
Rotation rx, ry, rz	Specify the rotation of the workobject.
Frame by points	Specify the frame position of the object frame.
Storage type	Select **PERS** or **TASK PERS**. Select the **Storage Type TASK PERS** if you intend to use the workobject in multimove mode.
Module	Select the module in which to declare the workobject.

8.8. Tooldata

Creating tooldata

1. In the **Layout** browser, make sure the robot in which to create the tooldata is set as the active task.
2. Click **Tooldata** to bring up a dialog box.
3. In the **Misc Data** group:
   - Enter the **Name** of the tool.
   - Select whether the tool is to be held by the robot in the **Robot holds tool** list.
4. In the **Tool Frame** group:
   - Define the **Position x, y, z** of the tool.
   - Enter the **Rotation rx, ry, rz** of the tool.
5. In the **Load Data** group:
   - Enter the **Weight** of the tool.
   - Enter the **Center of gravity** of the tool.
   - Enter the **Inertia** of the tool.
6. In the **Sync Properties** group:
   - In the **Storage type** list, select **PERS** or **TASK PERS**. Select **TASK PERS** if you intend to use the tooldata in MultiMove mode.
   - In the **Module** list, select the module in which to declare the tooldata.
7. Click **Create**. The tooldata appears as a coordinate system in the graphics window.

8.9 Target

8.9.1. Teach Target

Teaching a target

To teach a target, follow these steps:

1. In the **Layout** browser, select the workobject and tool for which you want to teach the target.
2. Jog the robot to the preferred position.

   NOTE: To jog a robot linearly, its VC must be running.
3. Click **Teach Target**.
4. A new target will be created in the browser, under the active workobject node. In the graphics window a coordinate system will be created at the TCP position. The configuration of the robot at the target will be saved.

8.9.2. Create Target

Creating a target

1. In the **Layout** browser, select the workobject in which you want to create the target.
2. Click **Create Target** to bring up a dialog box.
3. Select the **Reference** coordinate system you want to use to position the target:

If you want to position the target	Select
absolute in the world coordinate system of the station	**World**
relative to the position of the active workobject	**Work Object**
in a user-defined coordinate system	**UCS**

4. In the **Points** box, click **Add New** and then click the desired position in the graphics window to set the position of the target. You can also enter the values in the **Coordinates boxes** and click **Add**.
5. Enter the **Orientation** for the target. A preliminary cross will be shown in the graphics window at the selected position. Adjust the position, if necessary. To create the target, click **Create**.
6. If you want to change the workobject for which the target is to be created, expand the **Create Target** dialog box by clicking the **More** button. In the **WorkObject** list, select the workobject in which you want to create the target.
7. If you want to change the target name from the default name, expand the **Create Target** dialog box by clicking the **More** button and entering the new name in the **Target name** box.
8. Click **Create**. The target will appear in the browser and in the graphics window.

   NOTE: The created target will not get any configuration for the robot axes. To add the configuration values to the target, use either **ModPos** or the **Configurations** dialog box.

   NOTE: If using external axes, the position of all activated external axes will be stored in the target.

The Create Target dialog box

Reference	Select the reference coordinate system to which all positions or points will be related.
Position	Click in one of these boxes, and then click the position in the graphics window to transfer the values to the **Position** boxes.
Orientation	Specify the orientation of the target.
Add	Click this button to add a point and its coordinates to the **Points** list.
Modify	Click this button to modify an already defined point, after you have selected it in the **Points** list and entered new values.
Points	The target points. To add more points, click **Add New**, click the desired point in the graphics window, and then click **Add**.
More/Less	Click this button to expand or collapse parts of the create target dialog box.

Continues on next page

Target name	Here you can change the name of the target you are creating. It is visible only when the create target dialog box is expanded.
Workobject	Here you can change the workobject in which the target is to be created. It is visible only when the create target dialog box is expanded.

8.9.3. Create Jointtarget

Creating a jointtarget

1. Click **Create Jointtarget** to bring up a dialog box.
2. If you want to change the default name of the jointtarget, enter the new name in the **Name** box.
3. In the **Axes Values** group, do as follows:
   - For the **Robot axes**, click the **Values** box and then click the down arrow. The **Joint Values** dialog box will be displayed. Enter the joint values in the boxes and click **Accept**.
   - For the **Joint axes**, click the **Values** box and then click the down arrow. The **Joint Values** dialog box will be displayed. Enter the joint values in the boxes and click **Accept**.
4. Click **Create**. The jointtarget will appear in the browser and in the graphics window.

The Create Jointtarget dialog box

Name	Specify the name of the jointtarget.
Robot axes	Click the **Values** list, enter the values in the **Joint values** dialog box and click **Accept**.
External axes	Click the **Values** list, enter the values in the **Joint values** dialog box and click **Accept**.
Storage Type	Select the **Storage Type TASK PERS** if you intend to use the jointtarget in multimove mode.
Module	Select the module in which you want to declare the jointtarget.

8.10. Empty Path

Creating an empty path

1. In the **Paths&Targets** browser, select the folder in which you want to create the path.
2. Click **Empty Path**.
3. To set the correct motion properties for the targets, select the active process in the **Change Active Process** box in the **Elements** toolbar.
4. **NOTE**: If the active template is set to **MoveAbsJoint**:
    - A target that is dragged into a path will be converted into a **jointtarget** (recognized by a different icon on in the browser).
    - Jointtargets and their instructions can only use *wobj0* and *tool0*.
    - One target can not be used as different types, for example, **MoveJoint**, but must be deleted and re-created.
    - When the target has been synchronized with the virtual controller, the jointtarget values will be calculated and inserted in the RAPID program.

8.11. Path from Curve

Prerequisites

To create a path from a curve, the curve must first have been created in the station.

Creating a path from a curve

1. In the **Layout** browser, select the task in which you want to create the path.
2. Click **Path From Curve** to bring up a dialog box.
3. In the **Select curves from graphics** box, click **Add New** and then click the requested curve in the graphics window.
4. Select **Create on curve** or **Reference surface**:

To	Select
ensure that the path is generated on the curve. If cleared, the path may be generated on the reference surface, if one is used.	**Create on curve**
use a reference surface. The initial orientation of the target will be normal to the surface, and the parameters of the target will relate to this orientation.	**Reference Surface**

5. Select the workobject in which targets are to be created from the **Insert targets in workobject** list.
6. Set the **Target Parameters**:

To	Enter values in
specify the rotation around the X axis.	**Approach**
specify the rotation around the Y axis.	**Travel**
specify the rotation around the Z axis.	**Spin**
create a start target at the specified distance from the first target on the curve, in the approach direction.	**Approach**
create a departure target at the specified distance from the last target on the curve, in the approach direction.	**Depart**
start the path at a different location along the curve than the start point.	**Offset Start**
end the path at a different location along the curve than the end point.	**Offset End**
create an offset which will be the same at every target along the path.	**Local Target Offset**

Continues on next page

7. Set the **Approximation Parameters**:

To	Select or enter values in
specify the maximum allowed deviation between the chord of the curve and the path. A low tolerance will generate more targets than a high tolerance.	**Max chord dev**
select whether a linear motion shall represent the curve or if circular motion should be used in the path.	**Line/circular**
specify the minimum distance between the targets. This box is applicable only if a linear motion has been selected above.	**Min dist.**
select the maximum radius for circular motions in the path. Larger circular motions will be replaced by a linear motion. This box is applicable only if a circular motion has been selected above.	**Max rad.**

8. Click **Create**. The path and its targets appear in the browser and in the graphics window.

NOTE!

The targets created do not have any robot axis configurations set, which is required for use in programs with configuration monitoring. For information about setting robot axes configurations, see *Setting robot axis configuration for paths on page 87*.

The Create Path From Curve dialog box

Select curves from graphics	Select the curve that you want to create a path from by clicking **Add New** and then clicking the selected curve in the graphics window.
Create on curve	This check box ensures that the path is generated on the curve. If cleared, the path may be generated on the reference surface if one is used.
Reference Surface	This check box is selected when a reference surface is to be used. The initial orientation of the target will be normal to the surface, and the parameters of the target will relate to this orientation.
Insert targets in workobject	Select the workobject in which you want to create the path from the list.
Target Parameters	
Approach	Specifies the rotation angle around the X axis.
Travel	Specifies the rotation angle around the Y axis.
Spin	Specifies the rotation angle around the Z axis.
Approach	Creates a start target at the specified distance from the first target on the curve, in the approach direction.
Depart	Creates a departure target at the specified distance from the last target on the curve, in the approach direction.
Offset Start	If you want a different starting point than the beginning of the curve, enter the offset from the start point in the **Offset Start** box.
Offset End	If you want a different end point than the end of the curve, enter the offset from the end point in the **Offset End** box.

Continues on next page

Local Target Offset	Enter the local target offset to generate the same offset at each target.
Max Chord Dev	Specifies the maximum allowed deviation between the chord of the curve and the path. A low tolerance will generate a more accurate path with more targets than a high tolerance.
Linear/circular	Selects whether linear segments shall represent the curve or if circular segments should be used in the path.
Min Distance	Specifies the minimum distance between the targets. This box is only applicable for linear paths.
Max Rad	Specifies the maximum radius for circular movements in the path. Larger circular segments will be replaced by linear segments. This box is applicable only for circular paths.

8.12. MultiMove

Overview

For browsing between the pages of the MultiMove window, click the tabs in the navigation pane. By default the tabs are arranged in an order that corresponds to the typical workflow:

Setup tab

System Config	**Select System**	Here you select the system that contains the robots to program.
	System	Each robot in the system is presented in its own row in this grid. In the columns you make the settings as described below.
	Enable	Select this check box to use the robot in the MultiMove program.
	Type	Specify if the robot holds the tool or the work piece.
	Robot	Displays the name of the robot.
Path Config	**Update**	Click this button to update the paths in the grid if any of the paths have been changed. The button turns red if a change has been detected and an update is necessary.
	Paths	Each path in the station is presented in its own row in this grid. In the columns you make the settings as described below.
	Enable	Select this check box for the paths to use for the program.
	Order	Displays the order in which the paths will be executed. To change the order, use the lists in the path column for rearranging the rows in which the paths appear.
	Path	Sets the path to be executed here.
Start Position	**Select Robot that other shall jump to**	When creating a new start position, select a robot that the other will try to reach here.
	Apply	Jumps the other robots to the new start position.

Motion Behavior tab

This is used for specifying constraints and rules for how the robots shall move relative to each other. The default setting is no particular constraints, which results in the fewest joint movements. However, changing the motion behavior might be useful for:

- Locking the orientation or position of the tool.
- Optimizing cycle time or reachability by allowing tolerances.
- Avoiding collisions or singularity by restricting joint motions.

Continues on next page

Both Joint Influence and TCP Constraints restrict the robot's motions. Changes in these settings might result in lower performance or situations where it is impossible to find proper solutions. The weight values for Joint Weights and TCP Constraints set how much the setting for each joint or TCP direction shall affect the robots relative to each other. It is the difference between the weight values that matters, not the absolute values. If contradicting behaviors have been set, the one with the lowest weight value will win.

Tool Tolerance, instead of restricting, enables more motions. Therefore, tolerances may improve cycle and process times and enhance the reachability of the robots. Tolerances, too, have weight value; here is set how much the robots shall use the tolerance. A low value indicates that the tolerance will be used a lot, while a high value means that the robots will try to avoid using the tolerance.

The joint influence controls the balance of how much the robots will use their joints. Decreasing the weight value for one axis will restrict the motion for this axis, while increasing it will promote motion on this axis relative to alternative axes.

The TCP constraints control the position and orientation of the tool. Enabling a TCP constraint will decrease the motion of the tool and increase the motion of the work piece.

The tool tolerances control the allowed deviation between the tool and the work piece. By default, tolerances are not enabled, which means that no deviation is allowed. Enabling a tolerance, if applicable, might improve motion performance. For example, if the tool is symmetric around its Z axis, you can enable the Rz tolerance without affecting the accuracy of the generated paths.

The tool offset sets a fixed distance between the tool and the paths.

Joint Influence	**Select Robot**	Select the robot's joints to constrain in this box.
	Joints for Robot	Displays the robot's joints and their constraint weights. Each joint is presented in its own row.
	Axis	Displays which axis the constraint affects.
	Influence	Specify how much the motion for the axis is constrained. 0 means a locked axis, while 100 means no constraint relative to default constraint values.
TCP Constraints	**Active TCP**	This grid displays the position and rotations of the TCP together with their constraint weights.
	Enable	Select this check box to activate the constraint for this TCP pose.
	Pose	Displays the TCP pose that is affected by the constraint.
	Value	Specify the pose value to constrain at. Either type the value, or click the Pick from TCP button to use the values of the current TCP position.
	Influence	Specify how much the motion for the TCP value is constrained. 0 means a locked TCP at this pose, while 100 means no constraint relative to default constraint values.

Continues on next page

Tool Tolerance	Enable	Select this check box to activate the tolerance for this tool pose.
	Pose	Displays the tool pose that is affected by the constraint.
	Value	Specify the pose value to apply the tolerance around.
	Influence	Specify the size of the tolerance. 0 means no deviation is allowed, while 100 means all deviations are allowed.
Tool Offset	**Enable**	Select this check box to activate the offset for this tool pose.
	Pose	Displays the tool pose that is affected by the offset setting.
	Offset	Specify the value of the offset here.

Create Paths tab

This tab is used for creating RobotStudio paths for the MultiMove robots. The created paths will accord with the motions displayed during the most recently played test simulation.

With the settings group you set up the MultiMove properties that connect the tasks for the tool robot and work piece robot to each other.

With the WP robot settings group you set up the properties for the task that will be generated for the work piece robot.

The generate path group contains the button that creates the paths.

Settings	**Start ID**	Specify the first ID number for the synchronization of the instructions for the robots.
	ID step index	Specify the gap between the succeeding ID numbers.
	Sync ident prefix	Specify a prefix for the syncident variable, which connects the sync instructions in the tasks for the tool robot and the work piece robot with each other.
	Task list prefix	Specify a prefix for the tasklist variable, which identifies the tasks for the tool robot and work piece robot to synchronize.
	Path Prefix	Specify a prefix for the generated paths.
	Target Prefix	Specify a prefix for the generated targets.
WP robot settings	**WP Workobject**	Specify the work object to which the targets generated for the work piece robot shall belong.
	WP TCP	Specify which tool data the work piece robot shall use when reaching its targets.

Continues on next page

Continued

Generate Paths	**Create paths**	Clicking this button generates paths in RobotStudio for the latest tested motions according to the settings specified.

MultiTeach tab

With this tab you teach complete synchronized move instructions for the robots in the MultiMove program.

Settings	**Path Prefix**	Specify a prefix for the paths to create.
	Target Prefix	Specify a prefix for the generated targets here.
	Start ID	Specify the first ID number for the synchronization of the instructions for the robots.
	ID step index	Specify the gap between the succeeding ID numbers.
	Sync ident prefix	Specify a prefix for the syncident variable, which connects the sync instructions in the tasks for the tool robot and the work piece robot with each other.
	Task list prefix	Specify a prefix for the tasklist variable, which identifies the tasks for the tool robot and work piece robot to synchronize.
	Select type of Sync instruction	Select the type of synchronization to use. **Coordinated** implies that all move instructions are synchronized for the robots. **Semicoordinated** implies that the robots work independently at some times and wait for each other at other (like when repositioning the work piece). For detailed information about the coordination types, see the *Application manual - Multimove*.
	Setup	Select the robots for teaching targets. This grid also displays the workobjects and tools that will be used for the targets.
Teach	**MultiTeach Information**	Displays a hierarchal tree which contains the created move instructions. The tree is organized the same way as the tree in the Layout browser.
	MultiTeach	Creates move instructions for the robots selected in the settings to their current positions. The created move instructions are immediately inserted at their correct places in the MultiTeach Information tree.

Continues on next page

	Done	Confirms the creation of the instructions.

Test tab

RobotStudio's MultiMove window has a page with commands for testing multimove programs. Its default placement is at the bottom of the MultiMove window.

The status group displays the status of the simulation, that is, whether the current settings have been tested or if errors have occurred.

In addition to the status group, information from the virtual controller is also displayed in RobotStudio's Output window during simulation.

Play	**<< < > >>**	Jumps the robots, respectively, to the previous and next targets in paths. The double arrow buttons jump several targets at once, while the single arrow buttons jump one target for each click.
	Play	Click this button to move the robots along the paths. Play also has a list box in which you can activate the following commands: • **Save current position**: Saves the current start position. Since the calculated motions are based on the robot start position saving the start position is useful when testing alternative solutions. • **Restore saved position**: Moves the robots back to the saved start positions. • **Restore last closed loop position**: Moves the robots back to the list used start position. • **Close loop**: Finds a suitable start position based on the robots' current positions and prepares the calculation of movements. • **Calculate**: Calculates and executes the movements.
	Simulation speed	Sets the speed at which the simulation is performed.
Settings	**Stop at end**	Select this check box to make the simulation stop after running the paths one time. If cleared, the simulation will continue playing over and over until it is stopped manually.

Continues on next page

	Simulate Online	Select this check box to run the simulation as the movements are calculated. This is useful for troubleshooting purposes, since it displays and reports targets the robot cannot reach.
	Cancel on error	Select this check box to stop the simulation if an error occurs. Cancel on errors is recommended to use when simulate online is used to minimize the number of error messages once the first error is identified.
	Watch Close Loop	Select this check box to display the search for a suitable start position in the graphics window. Clear the check box to jump the robots to the start position when it is found.

The MultiMove configure system wizard

The MultiMove configure system wizard guides you through configuring robots and workobjects for MultiMove system. If the workobjects are not configured correctly when the MultiMove functions are started, you will be asked whether to run the wizard. You can also start it manually from the Tools page of MultiMove.

The wizard contains four pages, the information pane at the bottom indicates the current page.

Workpiece robot	The workpiece robot page contains a list in which you select the robot that holds the workpiece. Only one robot can be set up as workpiece robot. If your station has several robots that hold the workpiece, set up one of them as workpiece robot and the other as tools robots, and create paths for these robots in which they only hold the workpiece.
Tool robots	The tool robots page contains a list in which you select the robots that operate on the work piece. All robots selected as tools robots will be coordinated to the workpiece robot. Any robot of the system that is selected neither as workpiece robot or tool robot will not be coordinated.
Workobjects	The Workobjects page contains a box for each tool robot in which you specify the workobject in which the targets for the MultiMove paths shall be created. The wizard will attach this workobject to the workpiece robot, for enabling MutliMove. Either type in a name for a new Workobject to create in the box, or select the **Use existing WorkObject** check box and then select one from the list.
Result	The Result page displays a summary of the station configuration. Either click **Done** to finish or **Previous** to go back and change the setup.

Continues on next page

NOTE!

The wizard will not help you configure the RobotWare system correctly. If the correct options are not used you will not be able to synchronize generated MultiMove paths to the virtual controller, even if you can use the MultiMove functions in RobotStudio.

The Analyze path tool

This tool checks whether existing paths are coordinated correctly for MultiMove.

The analyzer opens in a window of its own and contains three pages. The information pane at the bottom indicates the current page.

Select Paths	**Enable**	Select to include the task in the analysis.
	Task	Displays the name of the task.
	Path	Select the path to analyze for the current task.
Analyze	**Analyze**	Click this button to start the analysis.
Report	 xx0600002648	OK. The paths are compatible in the specified aspect.
	 xx0600002649	Information. The paths are not fully compatible in the specified aspect, but the robot program may still be executable.
	 xx0600002650	Error. The paths are not compatible in the specified aspect, and the robot program is not executable.

The Recalculate ID tool

This is one of the tools for working manually with MultiMove programs. It sets new sync ID arguments to the move instructions in a MultiMove path. By using the tool with the same start ID and ID step index on all paths that shall be synchronized, you can be sure that the IDs match if all paths contain the same number of move instructions.

Title	Displays the name of the path to recalculate IDs for.
Start ID	Sets the number of the first ID in the path.
ID Step index	Sets the size of the step to increase the ID number for each move instruction.
Only update instructions that has ID defined	Select this check box to recalculate only those IDs for instructions that have existing IDs. Clear this check box to also create IDs for instructions that have no IDs (for example, if you have added new move instructions that shall be coordinated).
The **Only update instructions between SyncMove On/Off** check box.	Select this check box to affect only those move instructions that belong to already synchronized parts of the path. Clear this check box to update IDs for instructions in all parts of the path.

Continues on next page

The Convert path to MultiMove path tool

This is one of the tools for working manually with MultiMove programs. It adds sync ID arguments to all move instructions in the path and, optionally, `SyncMoveOn/Off` instructions, thus preparing an ordinary path for MultiMove use.

You use the tool on one path a time, so for creating a MultiMove program, you convert one path for each robot and then create a tasklist and syncidents which you add to the Sync instructions.

Title	Displays the name of the path to recalculate IDs for.
Start ID	Sets the number of the first ID in the path.
ID Step index	Sets the size of the step to increase the ID number for each move instruction.
Add SyncMove On/Off before and after	Select this check box to add instructions that start and stop the synchronization.

The Create Tasklist tool

This is one of the tools for working manually with MultiMove programs. It creates a variable of the RAPID data type `tasks`, which identifies the tasks that will be synchronized. In each `SyncMoveOn` or `WaitSyncTask` instruction, you then specify which tasklist to use.

Tasklist name	Specifies the name of the tasklist.
Tasks that will be included	Select the check box for each task to include in the list.

The Create Syncident tool

This is one of the tools for working manually with MultiMove programs. It creates a variable of the RAPID data type `SyncIdent`, which identifies the sync instructions that shall be synchronized.

Syncident name	Specifies the name of the `SyncIdent` variable to create.
Tasks that SyncIdent will be created in	Select the check box for each task to use this Syncident in.

8.13. Teach Instruction

Teaching a move instruction

1. In the **Layout** browser, make sure the settings for active robot, workobject, tool, motion type and path are appropriate for the move instruction to create.
2. Jog the robot to the desired location. If jogging the robot using the freehand mode, you can also use snap modes for snapping its TCP to objects in the station.
3. Click **Teach Instruction**. A move instruction is now created last in the path.

8.14. Move Instruction

Creating a move instruction and a corresponding target

To create a move instruction, follow these steps:

1. Click **Move Instruction** to bring up a dialog box.
2. Select the **Reference coordinate system** for the move instruction.
3. Enter the **Position** to reach for the move instruction by clicking **Add New** in the **Coordinates** box and then click the required to-points in the graphics window. You can also enter the values in the **Coordinates boxes** and click **Add**.
4. Enter the **Orientation** for the move instruction.
5. By clicking the **More/Less** button, you can expand or collapse the **Create Move Instruction** dialog box. When the dialog box has been expanded, you can change the **Target name** and the **Work object** to which the target (with the move instruction) will belong.
6. Click **Create** to create the move instruction. The move instruction will appear under the path node as a reference to the target.

The Create Move Instruction dialog box for jointtarget movements

Name	Here you can change the name of the target you create when creating the move instruction.
Robot axes	Specify the joint values for the robot. Select the box and click the list to set the values.
External axes box	Specify the joint values for external axes, if any exist in the station. Select the box and click the list to set the values.
Storage Type	Click this button to expand or collapse parts of the create move instruction dialog box.
Module	Specify the module in which the jointtarget shall be declared.

8.15. Action Instruction

Creating an action instruction

1. In the **Paths&Targets** browser, select where to insert the action instruction.

To insert the action instruction	Select
at the beginning of a path	the path
after another instruction	the proceeding instruction

2. Right click **Path** and select **Insert Action Instruction**.

   The **Create Action Instruction** dialog box appears.
3. From the **Instruction Templates** list, select the action instruction to create.
4. Optionally, modify instruction arguments in the **Instruction Arguments** grid. For detailed information about the arguments for each instruction, see the *Rapid reference manual*.
5. Click **Create**.

8.16. Instruction Template Manager

Importing a template

1. Click **Import** to bring up the **Open File** dialog box.
2. Select the file to import, and click **OK**.

Exporting a template

1. Select an exportable node in the tree view and click **Export** to bring up the **Save File** dialog box.
2. Click **OK**.

Validating the templates

1. Select a node in the tree view and click **Validate**.

   Any invalidity will be reflected by the icons and ToolTips of the respective node and reported in the Output window.

About the Instruction Template Manager

The Instruction Template Manager is used to add support for instructions other than the default set that comes with the RobotStudio.

For example, a robot controller system with the RobotWare Dispense option has specialized move instructions related to glueing like DispL and DispC. You can manually define the instruction templates for these using the Instruction Template Manager. The instruction templates are exported to XML format and reused later.

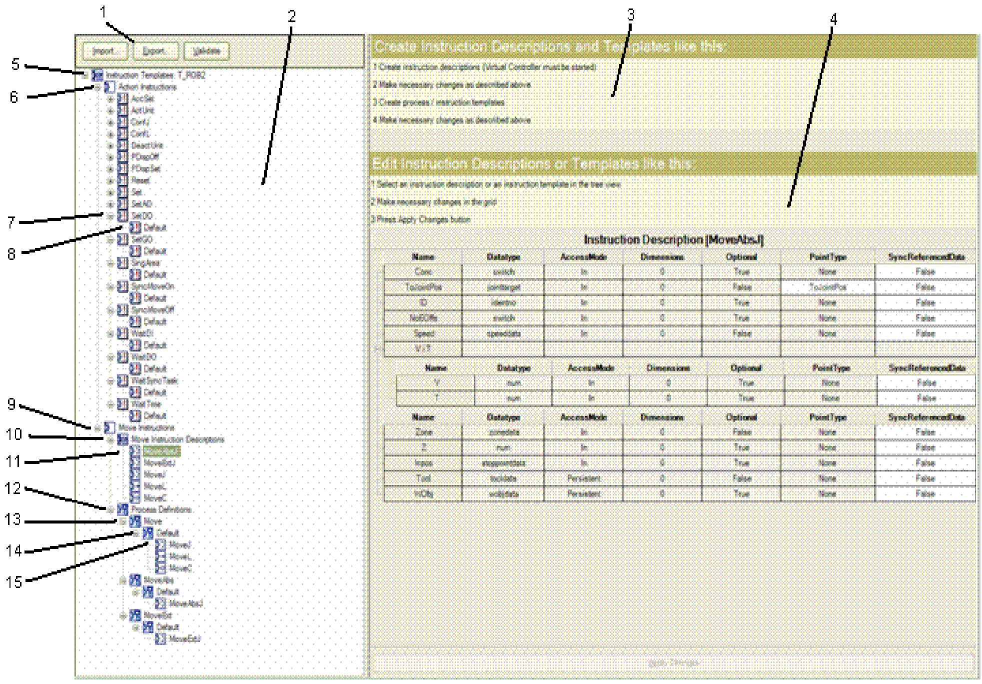
RobotStudio has pre-defined XML files that are imported and used for robot controller systems with the appropriate RobotWare options. These XML files has both the Move and Action instructions.

The instruction template supports the following Robotware options:

- Cap (Continuous Application Process)
- Disp (Dispense)
- Trigg (Fixed Position Events)
- Spot Pneumatic
- Spot Servo
- Spot Servo Equalizing
- Paint

Continues on next page

NOTE! It is recommended to use RobotStudio ArcWelding PowerPac while using RobotWare Arc.



xx0600003320

Item	Description
1	Buttons for importing, exporting and validating.
2	The instruction template tree. This hierarchal tree set organizes the templates. Templates are always the lowest level nodes. For details about specific nodes in the tree, see item 5 and below.
3	Brief description for editing and creating instruction templates.
4	The **Instruction grid**. All arguments and settings for the object selected in the tree are displayed here. Only white boxes are editable. Red values indicate that the values are invalid.
5	The **Instruction templates** top node. Here you can see to which task the templates belong.
6	The **Action instructions** node contains everything related to action instruction templates.
7	An **Action instruction description** node, here represented by the *Set DO* instruction, defines the arguments that can be set for the action instruction templates of that kind. You can create action instruction descriptions for all action instructions known by the system running on the virtual controller.
8	An **Action instruction template** node, here represented by *Default*, contains instances of the action instruction descriptions, with defined values for the arguments.
9	The **Move instructions** node contains everything related to move instruction templates.

Continues on next page

Continued

Item	Description
10	The **Move instruction descriptions** node contains all move instructions descriptions for the task. If the description for an instruction is not present in the list, right-click this node to add it. You can create move instruction descriptions for all move instructions known by the system running on the virtual controller.
11	A **Move instruction description** node, here represented by the *MoveAbsJ* node, defines the arguments that can be set for the move instruction templates of that kind. Unlike action instructions, instruction templates related to a certain move instruction descriptions are not stored in child nodes under the description, due to a more complex hierarchy.
12	The **Process definitions** node, which gathers all process definitions, contains sets of process templates which in turn contain instruction templates optimized for specific processes.
13	A **Process definition** node, here represented by the generic *Move* process, contains sets of process templates which in turn contain instruction templates optimized for specific processes.
14	A **Process template** node, here represented by the generic *Default* process, contains sets of move instruction templates with argument values optimized for specific processes. A process template can hold one move instruction template for each move instruction type defined by a move instruction description.
15	A **Move instruction template** node, here represented by *MoveJ*, contains instances of move instruction descriptions with argument values customized for specific processes.

8.17 The Freehand Group

8.17.1. Move

Moving an item

1. In the **Layout** browser, select the item you want to move.
2. Click **Move**.
3. In the graphics window, click one of the axes and drag the item into position.

8.17.2. Rotate

Rotating an item

1. In the **Layout** browser, select the item you want to rotate.
2. Click **Rotate**.
3. In the graphics window, click one of the rotational rings and drag the item into position.

   If you press the **ALT** key while rotating, the item will snap 10 degrees at a time.

8.17.3. Jog Joint

Jogging the joints of a robot

1. In the **Layout** browser, select the robot you want to move.
2. Click **Jog Joint**.
3. Click the joint you want to move and drag it to the preferred position.

   If you press the **ALT** key when jogging the joints of the robot, the robot will move 10 degrees at a time. If you press the **f** key, the robot will move 0.1 degree at a time.

8.17.4. Jog Linear

Jogging the TCP of a robot

1. In the **Layout** browser, select the robot you want to move.
2. In the **Freehand** group, click **Jog Linear**. A coordinate system will be displayed at the TCP of the robot.
3. Click the axis you want to move and drag the TCP to the preferred position.

   If you press the **f** key while jogging the robot linearly, the robot will move with a smaller step size.

8.17.5. MultiRobot Jog

Jogging robots in multirobot mode

1. In the **Freehand** group, click **MultiRobot Jog**. Select the robots to be jogged from the list of available robots.
2. Select the jogging mode, jog one of the robots and the other ones will follow the movement.

8.18. Viewpoint

Overview

A Viewpoint stores the location and direction of a virtual camera in the 3D environment. It stores points of interest in a station that are used create camera movements during simulation.

Creating Viewpoint

You can create a viewpoint in a station in two ways:

1. In the **Home** tab, click **View** and select **Create Viewpoint**.
2. In the **Layout** browser, right-click the station and select **Create Viewpoint**.

A viewpoint is created and displayed (as an eye icon) in the layout browser to the left.

NOTE! The position and direction of the Viewpoint can also be visualized as an arrow in the 3D graphics.

NOTE! By default, the newly created viewpoints are not visible and cannot be selected by clicking on the graphics.

Viewpoint functions

In the **Layout** browser, right-click **Viewpoint** to perform these functions:

Function	Description
Move to Viewpoint	Moves the active 3D view to the location stored in the viewpoint.
Update Viewpoint	Modifies the viewpoint to the current location and direction of the active 3D view.
Visible	Toggles the visibility of the viewpoint 3D representation.
Delete	Deletes the viewpoint.
Rename	Renames the viewpoint.

Move to Viewpoint

You can also move an active 3D view to the location stored in the Viewpoint using the Event manager.

1. Create a viewpoint. See *Creating Viewpoint on page 224*.
2. Add an event. See *Event Manager on page 258*.

   The Create new event dialog box appears.
3. Select **Simulation** under **Activation** and **Simulation time** under **Event trig type.** Click **Next.**
4. Set the activation time. Click **Next**.
5. Select **Move to Viewpoint** from **Set Actio type**. Click **Next**.
6. Select the viewpoint from **Select Viewpoint** and set the transition time.
7. Click **Finish.**

   NOTE! Move to viewpoint function is also executed when replaying the simulation in a Station Viewer.

9 The Modeling Tab

9.1. Overview

The Modeling tab

The Modeling tab contains the controls for creating and grouping components, creating bodies, measurements and CAD operations.

9.2. Component Group

Creating a component group

1. Click **Component Group**. The **Group** node will be displayed in the **Layout** browser.
2. Click the objects to add to the group. Drag them to the **Group** node.

9.3. Empty Part

Creating an empty part

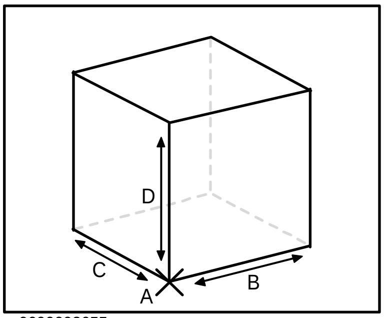
1. Click **Empty Part**. The **Part** node will be displayed in the **Layout** browser.

9.4. Solid

Creating a solid

1. Click **Solid** and then click the type of solid you want to create to bring up a dialog box.
2. Enter requested values in in the dialog box and click **Create**. For detailed information about the specific dialog box for the curve to create, see below:

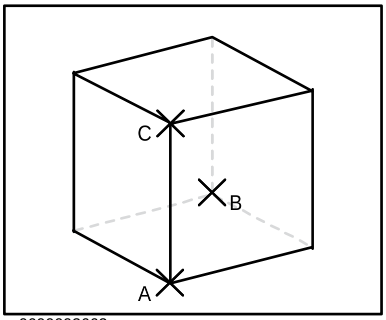
The Create Box dialog box



xx0600002655

Reference	Select the **Reference** coordinate system to which all positions or points will be related.
Corner Point (A)	Click in one of these boxes, and then click the corner point in the graphics window to transfer the values to the **Corner Point** boxes, or type the position. The corner point will be the local origin of the box.
Orientation	If the object shall be rotated relative to the reference coordinate system, specify the rotation.
Length (B)	Specify the box dimension along its X axis.
Width (C)	Specify the box dimension along its Y axis.
Height (D)	Specify the box dimension along its Z axis.

The Create Box from 3-Points dialog box



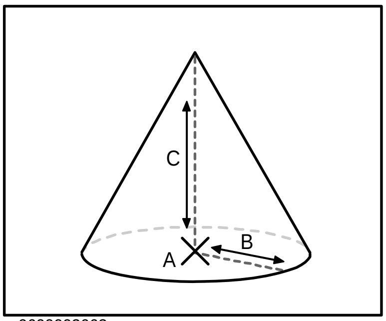
xx0600002662

Reference	Select the **Reference** coordinate system to which all positions or points will be related.
Corner Point (A)	This point will be the local origin of the box. Either type the position, or click in one of the boxes and then select the point in the graphics window.

Continues on next page

Point on diagonal of XY-plane (B)es	This point is the the corner, diagonal to the local origin. It sets the X and Y directions of the local coordinate system, as well as the dimension of the box along these axes. Either type the position, or click in one of the boxes and then select the point in the graphics window.
Indication Point Z-axis (C)	This point is the corner above the local origin, It sets the Z direction of the local coordinate system, as well as the dimension of the box along the Z axis. Either type the position, or click in one of the boxes and then select the point in the graphics window.

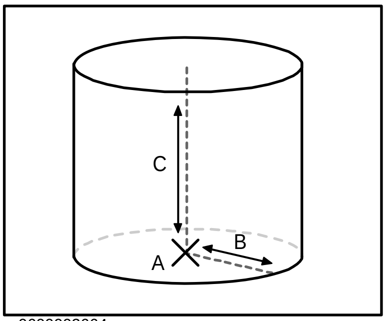
The Create Cone dialog box



xx0600002663

Reference	Select the **Reference** coordinate system to which all positions or points will be related.
Base Center Point (A)	Click in one of these boxes, and then click the center point in the graphics window to transfer the values to the **Base Center Point** boxes, or type the position. The center point will be the local origin of the cone.
Orientation	If the object shall be rotated relative to the reference coordinate system, specify the rotation.
Radius (B)	Specify the radius of the cone.
Diameter	Specify the diameter of the cone.
Height (C)	Specify the height of the cone.

The Create Cylinder dialog box



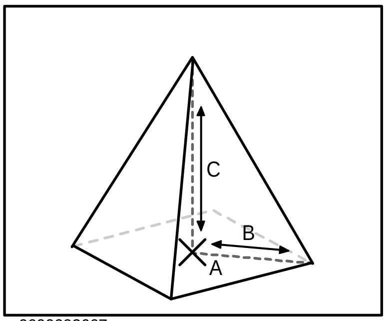
xx0600002664

Reference	Select the **Reference** coordinate system to which all positions or points will be related.

Continues on next page

Base Center Point (A)	Click in one of these boxes, and then click the center point in the graphics window to transfer the values to the **Base Center Point** boxes, or type the position. The center point will be the local origin of the cylinder.
Orientation	If the object shall be rotated relative to the reference coordinate system, specify the rotation.
Radius (B)	Specify the radius of the cylinder.
Diameter	Specify the diameter of the cylinder.
Height (C)	Specify the height of the cylinder.

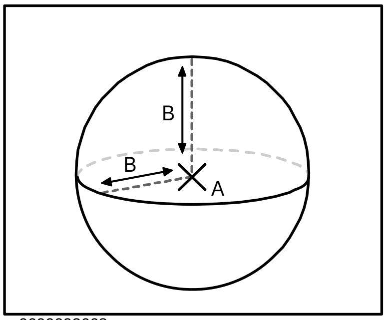
The Create Pyramid dialog box



xx0600002667

Reference	Select the **Reference** coordinate system to which all positions or points will be related.
Base Center Point (A)	Click in one of these boxes, and then click the center point in the graphics window to transfer the values to the **Base Center Point** boxes, or type the position. The center point will be the local origin of the pyramid.
Orientation	If the object shall be rotated relative to the reference coordinate system, specify the rotation.
Center to Corner Point (B)	Either type the position, or click in the box and then select the point in the graphics window.
Height (C)	Specify the height of the pyramid.
Number of Sides	Specify the number of sides of the pyramid. The maximum number of sides is 50.

The Create Sphere dialog box



xx0600002668

Reference	Select the **Reference** coordinate system to which all positions or points will be related.

Continues on next page

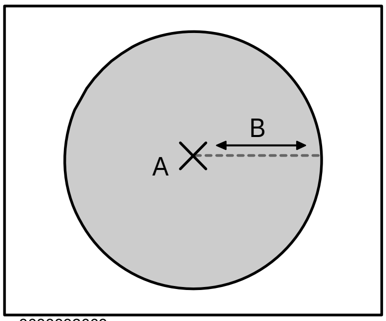
Center Point (A)	Click in one of these boxes, and then click the center point in the graphics window to transfer the values to the **Center Point** boxes, or type the position. The center point will be the local origin of the sphere.
Radius (B)	Specify the radius of the sphere.
Diameter	Specify the diameter of the sphere.

9.5. Surface

Creating a surface

1. Click **Surface** and then click the type of solid you want to create to bring up a dialog box.
2. Enter requested values in in the dialog box and click **Create**. For detailed information about the specific dialog box for the curve to create, see below:

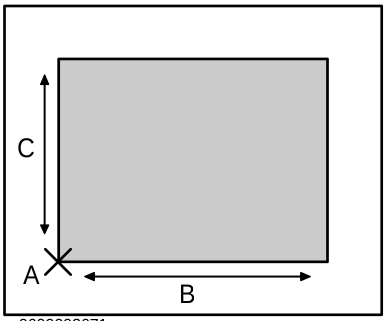
The Create Surface Circle dialog box



xx0600002669

Reference	Select the **Reference** coordinate system, to which all positions or points will be related.
Center Point (A)	Click in one of these boxes, and then click the center point in the graphics window to transfer the values to the **Center Point** boxes, or type the position. The center point will be the local origin of the circle.
Orientation	If the object shall be rotated relative to the reference coordinate system, specify the rotation.
Radius (B)	Specify the radius of the circle.
Diameter	Specify the diameter of the circle.

The Create Rectangle dialog box



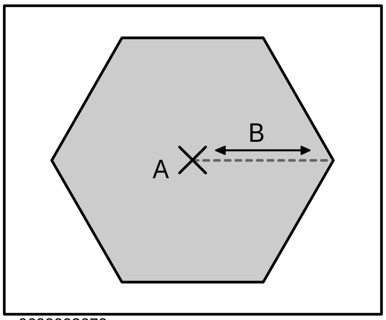
xx0600002671

Reference	Select the **Reference** coordinate system to which all positions or points will be related.
Start Point (A)	Click in one of these boxes, and then click the center point in the graphics window to transfer the values to the **Start Point** boxes, or type the position. The start point will be the local origin of the rectangle.
Orientation	If the object shall be rotated relative the reference coordinate system, specify the rotation.

Continues on next page

Length (B)	Specify the length of the rectangle.
Width (C)	Specify the width of the rectangle

The Create Surface Polygon dialog box



xx0600002670

Reference	Select the **Reference** coordinate system to which all positions or points will be related.
Center Point	Click in one of these boxes, and then click the center point in the graphics window to transfer the values to the **Center Point** boxes, or type the position. The center point will be the local origin of the polygon.
First Vertex Point	Either type the position, or click in one of the boxes and then select the point in the graphics window.
Vertices	Specify the number of the vertices here. The maximum number of vertices is 50.

The Create Surface from Curve dialog box

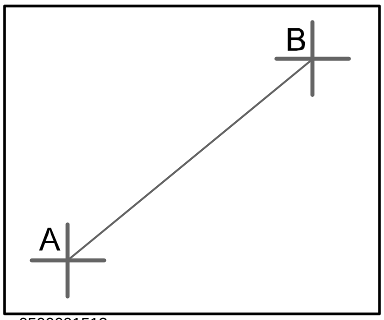
Select Curve from graphics	Select a curve by clicking it in the graphics window.

9.6. Curve

Creating a curve

1. Click **Curve** and then click the curve you want to create to bring up a dialog box.
2. Enter requested values in in the dialog box and click **Create**. For detailed information about the specific dialog box for the curve to create, see below:

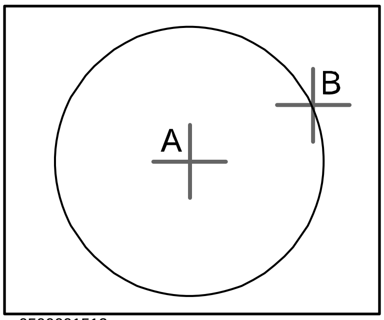
The Create Line dialog box



xx0500001513

Reference	Select the **Reference** coordinate system to which all positions or points will be related.
Start point (A)	Click in one of these boxes, and then click the start point in the graphics window to transfer the values to the **Start Point** boxes.
End Point (B)	Click in one of these boxes, and then click the end point in the graphics window to transfer the values to the **End Point** boxes.

The Create Circle dialog box

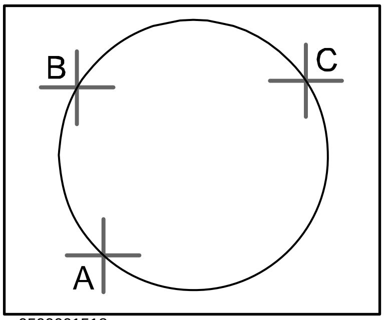


xx0500001512

Reference	Select the **Reference** coordinate system to which all positions or points will be related.
Center point (A)	Click in one of these boxes, and then click the center point in the graphics window to transfer the values to the **Center Point** boxes.
Orientation	Specify the orientation coordinates for the circle.
Radius (A-B)	Specify the radius of the circle.
Diameter	Alternatively, specify the diameter.

Continues on next page

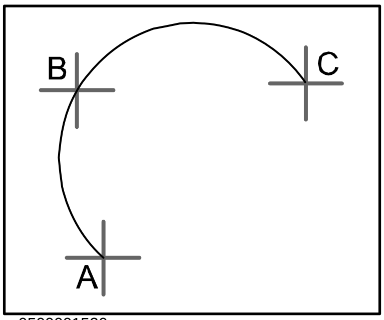
The Create Three Points Circle dialog box



xx0500001518

Reference	Select the **Reference** coordinate system to which all positions or points will be related.
First Point (A)	Click in one of these boxes, and then click the first point in the graphics window to transfer the values to the **First Point** boxes.
Second Point (B)	Click in one of these boxes, and then click the second point in the graphics window to transfer the values to the **Second Point** boxes.
Third Point (C)	Click in one of these boxes, and then click the third point in the graphics window to transfer the values to the **Third Point** boxes.

The Create Arc dialog box

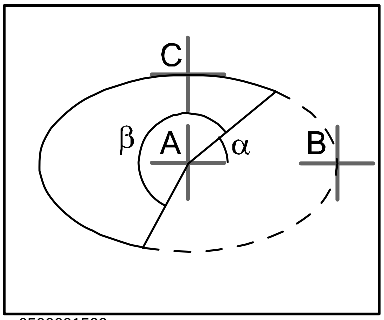


xx0500001520

Reference	Select the **Reference** coordinate system to which all positions or points will be related.
Start Point (A)	Click in one of these boxes, and then click the start point in the graphics window to transfer the values to the **Start Point** boxes.
Mid Point (B)	Click in one of these boxes, and then click the second point in the graphics window to transfer the values to the **Mid Point** boxes.
End Point (C)	Click in one of these boxes, and then click the end point in the graphics window to transfer the values to the **End Point** boxes.

Continues on next page

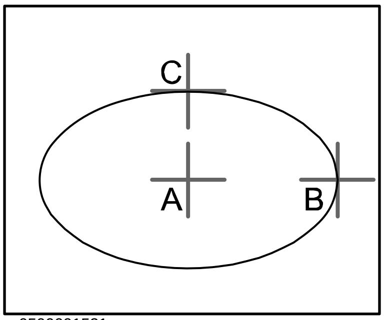
The Create Elliptical Arc dialog box



xx0500001522

Reference	Select the **Reference** coordinate system to which all positions or points will be related.
Center Point (A)	Click in one of these boxes, and then click the center point in the graphics window to transfer the values to the **Center Point** boxes.
Major Axis End Point (B)	Click in one of these boxes, and then click the end point for the major axis of the ellipse in the graphics window to transfer the values to the **Major Axis End Point** boxes.
Minor Axis End Point (C)	Click in one of these boxes, and then click the end point for the minor axis of the ellipse in the graphics window to transfer the values to the **Minor Axis End Point** boxes.
Start Angle (α)	Specify the start angle for the arc, measured from the major axis.
End Angle (β)	Specify the end angle for the arc, measured from the major axis.

The Create Ellipse dialog box

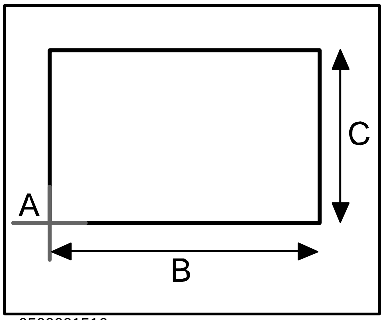


xx0500001521

Reference	Select the **Reference** coordinate system to which all positions or points will be related.
Center Point (A)	Click in one of the **Center Point** boxes, and then click the center point in the graphics window to transfer the values to the **Center Point** boxes.
Major Axis End Point (B)	Click in one of these boxes, and then click the end point for the major axis of the ellipse in the graphics window to transfer the values to the **Major Axis End Point** boxes.
Minor Radius (C)	Specify the length of the minor axis of the ellipse. The minor radius will be created perpendicular to the major axis.

Continues on next page

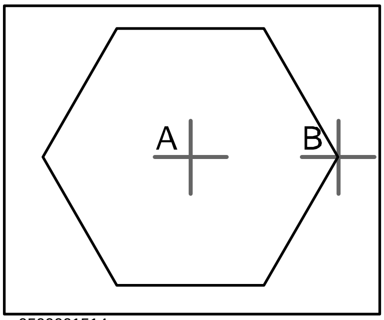
The Create Rectangle dialog box



xx0500001516

Reference	Select the **Reference** coordinate system to which all positions or points will be related.
Start Point (A)	Click in one of these boxes, and then click the start point in the graphics window to transfer the values to the **Start Point** boxes. The rectangle will be created in the positive coordinate directions.
Orientation	Specify the orientation coordinates for the rectangle.
Length (B)	Specify the length of the rectangle along the x axis.
Width (C)	Specify the width of the rectangle along the y axis.

The Create Polygon dialog box

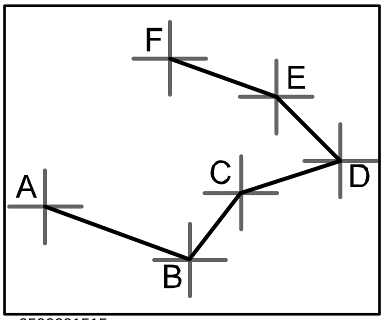


xx0500001514

Reference	Select the **Reference** coordinate system to which all positions or points will be related.
Center Point (A)	Click in one of these boxes, and then click the center point in the graphics window to transfer the values to the **Center Point** boxes.
First Vertex Point (B)	Click in one of these boxes, and then click the first vertex point in the graphics window to transfer the values to the **First Vertex Point** boxes. The distance between the center point and the first vertex point will be used for all vertex points.
Vertices	Specify the number of points to be used when creating the polygon. The maximum number of vertices is 50.

Continues on next page

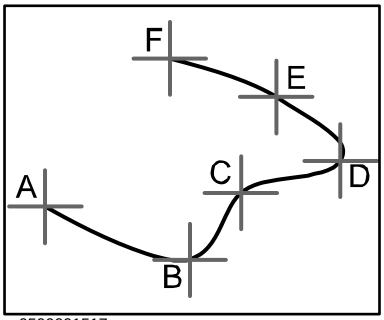
The Create Polyline dialog box



xx0500001515

Reference	Select the **Reference** coordinate system to which all positions or points will be related.
Point Coordinates	Specify each node of the polyline here, one at a time, by either typing the values, or by clicking in one of these boxes, and then selecting the point in the graphics window to transfer its coordinates.
Add	Click this button to add a point and its coordinates to the list.
Modify	Click this button to modify an already defined point, after you have selected it in the list and entered new values.
List	The nodes of the polyline. To add more nodes, click **Add New**, click the desired point in the graphics window, and then click **Add**.

The Create Spline dialog box



xx0500001517

Reference	Select the **Reference** coordinate system to which all positions or points will be related.
Point Coordinates	Specify each node of the spline here, one at a time, by either typing the values, or by clicking in one of these boxes, and then selecting the point in the graphics window to transfer its coordinates.
Ad	Click this button to add a point and its coordinates to the list.
Modify	Click this button to modify an already defined point, after you have selected it in the list and entered new values.
List	This nodes of the spline. To add more nodes, click **Add New**, click the desired point in the graphics window, and then click **Add**.

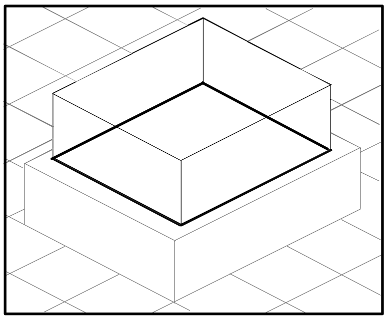
9.7. Border

Creating a curve

1. Click **Border** and then click the border you want to create to bring up a dialog box.
2. Enter requested values in in the dialog box and click **Create**. For detailed information about the specific dialog box for the border to create, see below:

The Create Border Between Bodies dialog box

To use the create border between bodies command, the station must contain at least two objects.

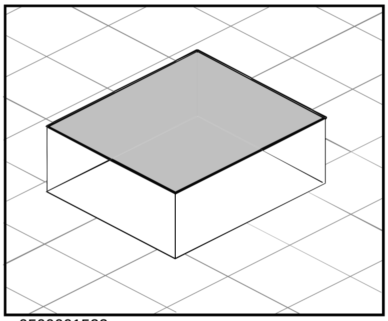


xx0500001524

First Body	Click in this box and then select the first body in the graphics window.
Second Body	Click in this box and then select the second body in the graphics window.

The Create Border Around Surface dialog box

To use the create border around surface command, the station must contain at least one object with a graphical representation.



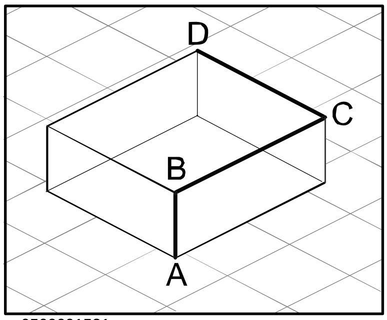
xx0500001523

Select Surface	Click in this box and then select a surface in the graphics window.

Continues on next page

The Create Border From Points dialog box

To use the create border from points command, the station must contain at least one object.

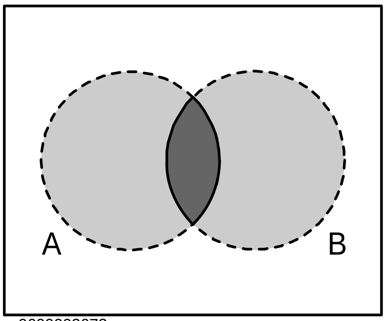


en0500001531

Selected Object	Click in this box and then select an object in the graphics window.
Point Coordinates	Specify the points that define the border here, one at a time, by either typing the values, or by clicking in one of these boxes, and then selecting the point in the graphics window to transfer its coordinates.
Add	Click this button to add a point and its coordinates to the list.
Modify	Click this button to modify an already defined point, after you have selected it in the list and entered new values.
List	The points that define the borders. To add more points, click **Add New**, click the desired point in the graphics window, and then click **Add**.

9.8. Intersect

The Intersect dialog box

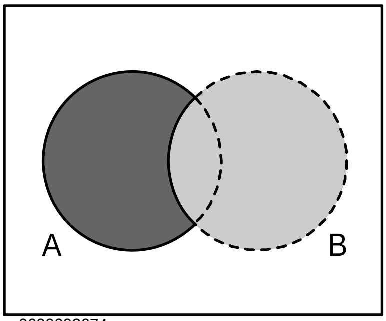


xx0600002673

Keep Original	Select this check box to keep the original bodies when creating the new body.
Intersect... (A)	Select the body from which you want to make an intersection (A) by clicking it in the graphics window.
...and (B)	Select the body with which you want to make an intersection (B) by clicking it in the graphics window. A new body will be created based on the common area between the selected bodies A and B.

9.9. Subtract

The Subtract dialog box

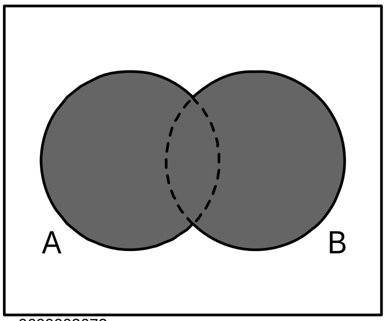


xx0600002674

Keep Original	Select this check box to keep the original bodies when creating the new body.
Subtract... (A)	Select the body from which you want to subtract (A) by clicking it in the graphics window.
...with (B)	Select the body you want to subtract (B) by clicking it in the graphics window. A new body will be created based on the area of body A subtracted with the common volume between body A and B.

9.10. Union

The Union dialog box



xx0600002672

Keep Original	Select this check box to keep the original bodies when creating the new body.
Union... (A)	Select the body that you want to unify (A) by clicking it in the graphics window.
...and (B)	Select the body that you want to unify (B) by clicking it in the graphics window. A new body will be created based on the areas of the two selected bodies A and B.

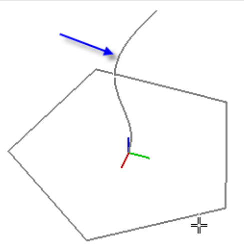
9.11. Extrude Surface or Curve

Extruding a surface or curve

1. From the selection level toolbar, select **Surface** or **Curve**, as appropriate.
2. In the graphics window, select the surface or curve you wish to extrude. Click **Extrude Surface** or **Extrude Curve**, as appropriate. The **Extrude Surface or Curve** dialog box opens below the **Modeling** browser.
3. For extrusion along a vector, fill in the values.

   For extrusion along a curve, select the **Extrude Along Curve** option. Click the **Curve** box, and select the curve in the **Graphics** window.
4. If you wish the form to appear as a surface model, clear the **Make Solid** check box.
5. Click **Create**.

The Extrude Surface or Curve dialog box

Surface or Curve	Denotes the surface or curve to be extruded. To select the surface or curve, first click in the box, then select the surface or curve in the graphics window.
Extrude Along Vector	Enables extrusion along a specified vector.
From Point (mm)	The start point of the vector.
To Point (mm)	The end point of the vector.
Extrude Along Curve	Enables extrusion along a specified curve.
Curve	Denotes the curve used as a sweep path.  xx0600003076 To select the curve, first click the box, then the curve in the graphics window.
Make Solid	Select this check box to convert the extruded form into a solid.

9.12. Line from Normal

Creating a line from normal

1. Click **Surface Selection**.
2. Click **Line to Normal** to bring up a dialog box.
3. Click on a face to select it in the **Select Face** box.
4. In the **Length** box, specify a length for the line.
5. Optionally, select the **Invert Normal** check box to invert the direction of the line.
6. Click **Create**.

9.13. The Measure Group

TIP!

Make sure to select the appropriate snap mode and selection levels before making your measurements.

Measuring distances or angles

1. Click the type of measurement you want to use:

To measure the	Select
distance between two points you select in the graphics window.	**Point to point**
angle defined by three points you select in the graphics window. The first point to select is the converging point, thereafter you shall select one point on each line.	**Angle**
diameter, with the circle defined by three points you select from the graphics window.	**Diameter**
closest distance between two objects you select in the graphics window.	**Minimum distance**

   The mouse pointer will turn into a ruler when you have activated any of the measurement functions.

2. In the graphics window, select the points or objects to measure between. Information about the measuring points is displayed in the **Output** window.

   The results will be displayed in the **Measurements** tab in the **Output** window when all points have been selected.

3. Optionally, repeat step 3 for making a new measurement of the same type.

TIP!

You can also activate and deactivate the measurement functions from the measurement toolbar.

9.14. Create Mechanism

Create a new mechanism

1. Click **Create Mechanism.**

   The Mechanism Modeler opens in create mode.
2. In the **Mechanism Model Name** box, enter a mechanism name.
3. From the **Mechanism Type** list, select a mechanism type.
4. In the tree structure, right-click **Links**, and then click **Add Link** to bring up the **Create Link** dialog box.

   A suggested name appears in the **Link Name** box.
5. In the **Selected Part** list, select a part (which will be highlighted in the graphics window) and click the arrow button to add the part to the Parts list box.

   The **Selected Part** list then automatically selects the next part, if any more are available. Add these, as required.
6. Select a part in the Parts list box, enter any values in the **Selected Parts** group boxes, and then click **Apply to Part**.

   Repeat for each part, as required.
7. Click **OK**.
8. In the tree structure, right-click **Joints**, and then click **Add Joint** to bring up the **Create Joint** dialog box.

   A suggested name appears in the **Joint Name** box.
9. Complete the **Create Joint** dialog box, and then click **OK**.
10. In the tree structure, right-click **Frame/Tool Data**, and then click **Add Frame/Tool** to bring up the **Create Frame/Tool** dialog box.

    A suggested name appears in the **Frame/Tool Data name** box.
11. Complete the **Create Frame/Tool** dialog box, and then click **OK**.

    The validity criteria for the **Frame/Tool** node are as follows:
12. In the tree structure, right-click **Calibration**, and then click **Add Calibration** to bring up the **Create Calibration** dialog box.
13. Complete the **Create Calibration** dialog box, and then click **OK**.
14. In the tree structure, right-click **Dependency**, and then click **Add Dependency** to bring up the **Create Dependency** dialog box.
15. Complete the **Create Dependency** dialog box, and then click **OK**.
16. If all nodes are valid, compile the mechanism, see *Compiling a mechanism on page 248*.

Create Conveyor mechanism

1. Click **Create Mechanism.**

   The Mechanism Modeler opens in create mode.
2. In the **Mechanism Model Name** dialog box, enter a mechanism name.
3. From the **Mechanism Type** list, select **Conveyor**.
4. From the **Selected Part list**, select **Part.**

Continues on next page

5. In the **Position of Calibration frame** list, enter the base frame values relative to the local origin of the selected graphic component.
6. In the **Conveyor length** box, enter the length of the conveyor.

   The **Compile Mechanism** button is enabled.
7. In the **Attachment Points** box, set the **Pitch** and **Count** value.
8. Click **Add** to create new attachment points.
9. Click **Compile Mechanism,** to compile the mechanism. See *Compiling a mechanism on page 248*.
10. In the **Layout** browser, right-click the conveyor mechanism and select **Save As Library.** Close the station.
11. Build a new system. See *Building a new system on page 142*.

    NOTE! On the **Modify Options** page of the **System Builder**, scroll down to the **Motion coordination part 3** group and select **606-1 Conveyor Tracking** check box.
12. Create new station using this new system. See *Robot System on page 188*.

    NOTE! After starting the system, when asked to select the library for the conveyor mechanism browse and select the already saved library.

Compiling a mechanism

When compiling, a new mechanism, created in the create mode of the Mechanism Modeler, is added to the station with the default name "Mechanism_" followed by an index number.

When compiling, an existing editable mechanism, modified in the modify mode of the Mechanism Modeler, is saved without any poses, joint mapping ot transition times.

To compile a mechanism, follow these steps:

1. To compile a new or edited mechanism, click **Compile Mechanism**.

   The mechanism is inserted into the active station. The link parts are cloned with new names, but the corresponding links will update their part references. When the Mechanism modeler is closed, these cloned parts will be removed.
2. The Mechanism Modeler now switches to modify mode. To complete the mechanism, see below.

Completing or modifying a mechanism

To complete the modeling of a mechanism, follow these steps:

1. If the values in the **Joint Mapping** group are correct, click **Set**.
2. Configure the Poses grid. To add a pose, click **Add** and then complete the **Create Pose** dialog box. Click **Apply**, followed by **OK**.

   To add a pose, click **Add** and then complete the **Create Pose** dialog box. Click **Apply**, followed by **OK**.

   To edit a pose, select it in the grid, click **Edit**, and then complete the **Modify Pose** dialog box. Click **OK**.

   To remove a pose, select it in the grid and then click **Remove**.
3. Click **Edit Transition Times** to edit transition times.
4. Click **Close**.

Continues on next page

The Create Mechanism dialog box

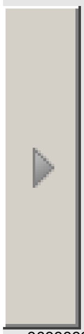
Mechanism Model Name	Specifies the model name of the mechanism.
Mechanism Type	Specifies the mechanism type.
Tree structure	The components of the mechanism in a tree structure. The tree structure will not be visible unless the mechanism is editable. Each node (link, joint, frame, calibration and dependency) can be edited in its own dialog box, see below.
Compile Mechanism	Click this button to compile the mechanism. This button will not be visible unless the mechanism is editable and the mechanism model name is valid.

The Create Conveyor Mechanism dialog box

Mechanism Model Name	Specifies the model name of the conveyor mechanism.
Mechanism Type	Specifies the diffrent mechanism types.
Selected Part	Specifies the part to be selected for the conveyor.
Position of Calibration frame	Specifies the baseframe value relative to the local origin of the selected graphic component.
Conveyor Length	Specifies the length of the conveyor.
Attachment Points	Specifies the conveyor position to attach the workpieces.
Compile Mechanism	Click this button to compile the mechanism. This button will not be visible unless the mechanism is editable and the mechanism model name is valid.

The Create/Modify Link dialog box

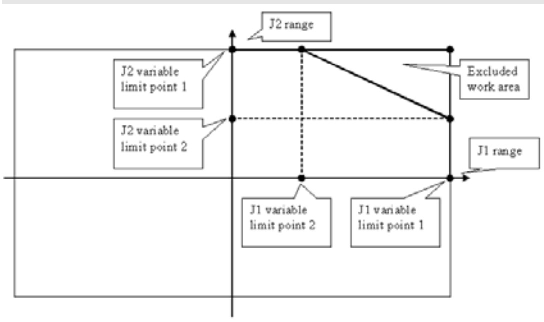
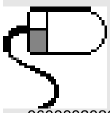
A link is a moving component of a mechanism. Selecting a link node will highlight it in the graphics window.

Link Name	Specifies the name of the link.
Selected Part	Specifies the parts to add to the Part list box.
Set as BaseLink	The BaseLink is where the kinematical chain begins. This must be the parent of the first joint. A mechanism may have only one BaseLink.
 xx0600003086	Adds a selected part to the Part list box.
Remove Part	Click this button to remove a selected part from the Part list box.
Selected Part	This group manipulates the transform of the selected part.
Part Position	Specify the position of the part.
Part Orientation	Specify the orientation of the part.
Apply to Part	Click this button to apply the settings to the part.

Continues on next page

The Create/Modify Joint dialog box

A joint is the axis where two links move in relation to each other, rotationally or prismatically. Selecting a joint node will display a yellow-green line in the graphics window.

Joint Name	Specifies the name of the joint.
Joint Type.	Specifies the joint type. The default option is **Rotational**. Changing the Joint Type clears the Joint Limits below.
Parent Link	Specifies the parent link, usually the first joint of the mechanism.
Child Link	Specifies the child link. To be valid, the parent and child links may not be identical to each other, and the pair must be unique.
Active	Select this check box to make the joint active. An active joint is one that a user can move, while an inactive joint is a slave to an active joint.
Joint Axis	This group specifies the axis around or along which the child moves.
First Position	Specify the start point of the axis vector.
Second Position	Specify the end point of the axis vector.
Jog Axis	Demonstrates how the child link moves along its axis.
Limit Type	Specifies the limits in each direction to which a joint may move. The options are **Constant**, **Variable** and **No**.
Joint Limits	This group is visible in **Constant** or **Variable** mode.
Min Limit	Specifies the minimum joint limit.
Max Limit	Specifies the maximum joint limit.
Joint Limits	This group is visible in **Constant** or **Variable** mode.
Variable Limits	In **Variable** mode, variable limit points may be added as an advanced way of delimiting the area of movement.  xx0600003106
 xx0600003086	Adds a selected point to the Point list box.
Remove	Click this button to remove a selected point from the Point list box.

The Modify Frame/Tool Data dialog box

A frame/tool data node determines the link and location of a frame.

Frame/Tool Data name	Specifies the name of the frame or tool data.
Belongs to Link	Specifies the link to which the frame or tool belongs.

Continues on next page

Position	Specify the position of the transform.
Orientation	Specify the orientation of the transform.
Select values from target/ Frame	Select this box to select the values from a target or frame, which is selected in the box beneath the check box.
Tool Data	This group is visible if the mechanism is a tool.
Mass	Specifies the mass of the tool.
Center of Gravity	Specify the center of gravity of the tool.
Moment of Inertia Ix, Iy, Iz	Specify the moment of inertia of the tool.

The Create Calibration dialog box

A calibration contains transforms for calibrating the joints. Two calibrations cannot share the same joint.

Calibration belongs to Joint	Specifies the joint to be calibrated.
Position	Specify the position of the transform.
Orientation	Specify the orientation of the transform.

The Create Dependency dialog box

A dependency is a relationship between two joints, by either a factor or a complex formula.

Joint	Specifies the joint whose motion will be controlled by other joints.
Use LeadJoint and factor	Select this option to specify a lead joint and factor.
LeadJoint	Specifies the lead joint.
Factor	This list holds a double which denotes the extent to which the lead joint will control the main joint.
Use Formula	Select this option to enter a formula in the box.

The Modify Mechanism dialog box

The **Modify Mechanism** dialog box contains the objects found in the **Create mechanism** dialog box, as well as the following:

Joint Mapping	These boxes handle the joint mapping of the mechanism. When editing, the mechanism must be disconnected from its library. The values must be integers from 1 – 6 in ascending order.
Set	Click this button to set the joint mapping.
Poses	Displays the poses and their joint values. Selecting a pose will move the mechanism to it in the graphics window.
Add	Click this button to bring up the **Create Pose** dialog box for adding a pose.
Edit	Click this button to bring up the **Modify Pose** dialog box for editing a selected pose. A SyncPose cannot be edited unless the mechanism is disconnected from its library.
Remove	Click this button to remove the selected pose. A single SyncPose cannot be removed.

Continues on next page

Set Transition Times	Click this button to edit the transition times.

The Create/Modify Pose dialog box

Pose Name	Specifies the name of the pose. If the pose is a SyncPose, this box is not editable. The names "HomePosition" and "SyncPosition" are disallowed.
Home Pose	Select this box to specify the home pose of the mechanism. If selected, the non-editable pose name will be "HomePose".
Launch Joint Jog Tool	Click this button to bring up the joint jog tool.
Use Current	Click this button to set the current joint values in the **Joint Values** group.
Reset Values	Click this button to reset the joint values in the **Joint Values** group to what they were when the dialog box was opened.
Joint Values	Specify the joint values of the pose.

The Set Transition Times dialog box

The **Set Transition Times** dialog box is designed like a distance table in a road atlas. The default values are zero.

From Pose	Specifies the start of the transition for the named pose.
To Pose	Specifies the end of the transition for the named pose.

9.15. Create Tool

Creating a tool

You can create a robot hold tool by using the **Create Tool Wizard**. The wizard allows you to easily create a tool from an existing part or by using a dummy part to represent a tool. To create a tool complete with tooldata, follow these steps:

1. Click **Create Tool**.
2. In the **Tool Name** box, enter a tool name and choose one of the following options:

Option	Action
Use Existing	Select one of the existing parts from the list. The selected part will represent the tool graphics. The selected part must be a single part. Parts with attache-ments cannot be selected.
Use Dummy	A cone will be created to represent the tool.

3. Continue entering the **Mass** of the tool, the **Center of Gravity** and the **Moment of Inertia Ix, Iy, Iz**, if these values are known. If you do not know the correct values, the tool can still be used for programming motions, but this data must be corrected before running the program on real robots or measuting cycle times.

   Tip! If the tool is built from materials with a similar density, you can find the center of gravity by clicking the tool model using the **Center of gravity** snap mode.
4. Click **Next** to go the next page of the wizard.
5. In the **TCP Name** box, enter a name for the Tool Center Point (TCP). The default name is the same as the name of the tool. If creating several TCPs for one tool, each TCP must have a unique name.
6. Enter the position of the TCP relative to the world coordinate system, which represents the tool mounting point, by any of the methods below:

Method	Description
Read values from existing target or frame	Click in the **Values from Target/Frame** box, then select the frame either in the graphics window or the **Paths&Targets** browser.
Enter position and orientation manually.	In the **Position** and **Orientation** boxes, type the values. **NOTE**: If **Use Dummy Part** is selected, the position value can not be 0,0,0. At least one coordinate has to be > 0 in order for a cone to be created.

7. Click the arrow right button to transfer the values to the **TCP(s):** box.

   If the tool shall have several TCPs, repeat steps 5 to 7 for each TCP.
8. Click **Done**. The tool will be created and appear in the **Objects** browser and in the graphics window.

Creating tooldata for an existing geometry

To create tooldata for an existing geometry, follow these steps:

1. Make sure the robot in which to create the tooldata is selected.
2. Click **Create Tool** and select **Use Existing** and the imported tool from the list.

Continues on next page

3. Enter the requested data in the boxes in the **Create Tool Wizard**.
4. Attach the tool by dragging it to the robot.

What to do next

To make the tool ready to use, do one of the following:

- To make the robot hold the tool, attach the tool to the robot.
- In the graphics window, check the position and orientation of the TCP. If it is incorrect, modify the values in the tool frame part of the tooldata.
- To simplify future usage of the created tool, save it as a library. On the **File** menu, click **Save As Library**. Browse to the folder where you want to store the tool component, enter a name for the tool component and click **Save**.

10 The Simulation Tab

10.1. Overview

The Simulation tab

The Simulation tab contains the controls for setting up, configuring, controlling, monitoring and recording simulations.

10.2. Create Collision Set

Overview

A collision set contains two groups, *Objects A* and *Objects B*, in which you place the objects to detect any collisions between them. When any object in *Objects A* collides with any object in *Objects B*, the collision is displayed in the graphical view and logged in the output window. You can have several collision sets in the station, but each collision set can only contain two groups.

Creating a collision set

1. Click **Create Collision Set** to create a collision set in the **Layout** browser.
2. Expand the collision set and then drag one of the objects to the **ObjectsA** node to check for collisions.

   If you have several objects you want to check for collisions with objects in the **ObjectsB** node, for example, the tool and the robot, drag all of them to the **ObjectsA** node.
3. Drag the objects to the **ObjectsB** node to check for collisions.

   If you have several objects you want to check for collisions with objects in the **ObjectsA** node, for example, the work piece and the fixture, drag all of them to he **ObjectsB** node.

10.3. Simulation Setup

Prerequisites

To set up a simulation, the following conditions must be met:

- At least one path must have been created in the station.
- The paths to be simulated must have been synchronized to the virtual controller.

Setting up a simulation

1. Click **Simulation Setup** to bring up a dialog box.
2. In the **Select Active Tasks** box, select the tasks to be active in the simulation.
3. If a single controller has been selected, select either the **Continuous** or **Single Cycle** option.
4. From the **Available Procedures** list, transfer the procedures to be active in the simulation to the **Main Sequence** lists by selecting them and clicking the left arrow button between the lists. (This creates a procedure call in the main procedure).
5. To start the simulation from another procedure than the actual Main procedure (that is, to set a temporary Main procedure instead of overwriting the current one), click **Select entry point** and then specify the procedure to use as Main.
6. Click **Apply** to set the simulation. If you click **OK**, the simulation will be set and the dialog box will be closed.

NOTE!

If the simulation uses events or involves several different controllers, the virtual time mode **Time Slice** shall be used to make sure that the timing between the controllers is simulated correctly. For more information about virtual time modes, see *Simulation Overview on page 117*.

The Setup Simulation dialog box

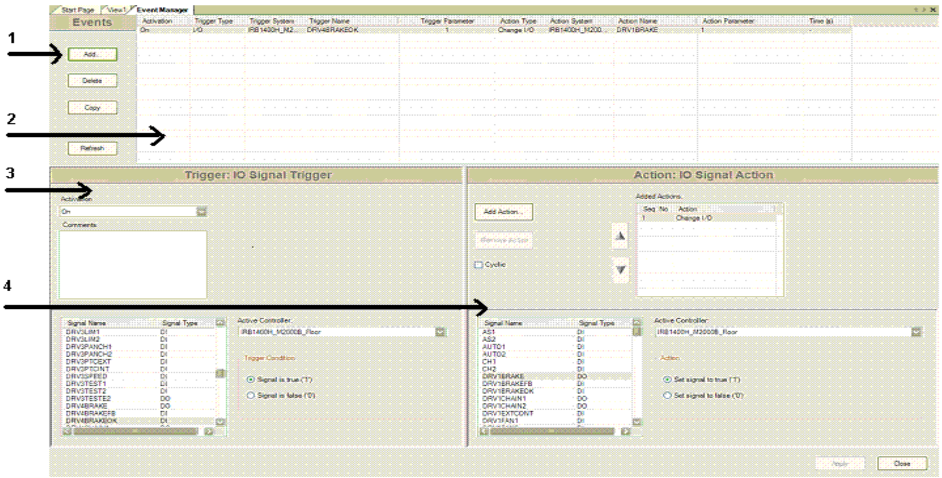
Select Active Tasks	Select the tasks to be active in the simulation.
The Run Mode options	Select the run mode when a single controller is selected. The options are **Continuous** and **Single Cycle**.
Main Sequence	Displays the procedures that will be run when you play the simulation.
<-	Click the arrow left button to transfer the selected procedure to the **Main Sequence** box.
X	Click this button to remove the selected procedures or sequences from the **Main sequence** box.
arrow up	Click the arrow up button to move the sequence up in the list in the **Main Sequence** box or in the **Available Procedures** box.
arrow down	Click the arrow down button to move the sequence down in the list in the **Main Sequence** box or in the **Available Procedures** box.
Available Procedures	Displays all procedures available in the controller.
Select entry point	Click this button to set up several simulations at the same time. In the dialog box, enter a new name for the main procedure, and select the procedures to be included in the new simulation. Click **OK**.

10.4. Event Manager

Creating an event

1. Click **Event Manager**.
2. Click **Add** to open the New Event Wizard.
3. Complete the New Event wizard to create the event.

Event manager main parts



xx0600003098

Part	Description
1	The **Task** pane. Here you create new events, or copy or delete existing events selected in the Event grid.
2	The **Event** grid. Displays all events in the station. Here you select events to edit, copy or delete.
3	The **Trigger** editor. Here you edit the properties of the events trigger. The upper part of the trigger editor is the same for all triggers, and the lower part adapts to the selected trigger type.
4	The **Action** editor. Here you edit the properties of the events action. The upper part of the action editor is the same for all actions, and the lower part adapts to the selected action type.

The task pane parts

Part	Description
Add	Starts the **Create New Event Wizard**.

Continues on next page

Part	Description
Delete	Deletes the event selected in the Event grid.
Copy	Copies the event selected in the Event grid.
Refresh	Refreshes the Event manager.

The event grid columns

In the event grid, each row is a an event and the columns in the grid display their properties.:

Column	Description
Activation	Displays whether the event is active or not. **On** = The action is always carried out when the trigger event occurs. **Simulation** = The action is only carried out if the trigger event occurs when running a simulation. **Off** = The action is not carried out when the trigger event occurs.
Trigger Type	Displays the type of condition that triggers the action. **I/O Signal Trigger** = Changes a digital I/O signal. **I/O Connection** = Simulates the behavior of a Programmable Logic Controller (PLC). **Collision Trigger** = Starts or ends a collision or near-miss between objects in a collision set. The trigger type cannot be changed in the trigger editor. If you want another trigger type than the current one, create a completely new event.
Trigger System	When the trigger type is *I/O Signal Trigger*, this column displays to which system the signal used as trigger belongs. A dash (-) signifies a virtual signal.
Trigger Name	The name of the signal or collision set used as trigger.
Trigger Parameter	Displays the condition of the event under which triggering occurs. **0** = The I/O signal used as trigger switches to false. **1** = The I/O signal used as trigger switches to true. **Started** = A collision starts within the collision set used as trigger. **Ended** = A collision ends within the collision set used as trigger. **Near miss started** = A near-miss starts within the collision set used as trigger. **Near miss ended** = A near-miss ends within the collision set used as trigger.

Continues on next page

Column	Description
Action Type	Displays the action type that occurs in conjunction with the trigger **I/O Signal Action** = Changes the value of a digital input or output signal. **Attach Object** = Attaches an object to another. **Detach Object** = Detaches an object from another. **Turn On/Off Simulation Monitor** = Toggles the simulation monitor of a specific mechanism. **Turn On/Off Timer** = Toggles the process timer. **Move Mechanism to Pose** = Moves the selected mechanism to a predefined pose and thereafter sends a station signal. Activates or deactivates the process timer. **Move Graphical Object** = Moves a graphical object to a new position and orientation. **Show/Hide Graphical Object** = Shows or hides the graphical object. **Do Nothing** = No action occurs. **Multiple** = The event triggers multiple actions, either all at once or one at a time, each time the trigger is activated. Each action can be viewed in the action editor.
Action System	When the action type is *Change I/O*, this column displays the system to which the signal to change belongs. A dash (-) signifies a virtual signal.
Action Name	Displays the name of the signal to change, when the action type is *Change I/O*.
Action Parameter	Displays the condition after the action has occurred. **0** = The I/O signal will be set to false. **1** = The I/O signal will be set to true. **On** = Turns the process timer on. **Off** = Turns the process timer off. **Object1 -> Object2** = Displays the object to which another will be attached when the action type is Attach object. **Object1 <- Object2** = Displays the object from which another will be detached when the action type is Detach object. **Ended** = A collision ends within the collision set used as trigger. **Near miss started** = A near-miss starts within the collision set used as trigger. **Near miss ended** = A near-miss ends within the collision set used as trigger. **Multiple** = Signifies multiple actions.
Time	Displays the time when the event trigger was executed.

Continues on next page

The trigger editor parts

In the trigger editor you set the properties of the trigger. The upper part of the editor is common for all types of triggers, and the lower part adapts to the trigger type at hand.

Parts common to triggers

Part	Description
Activation	Sets whether the event is active or not. **On** = The action is always carried out when the trigger event occurs. **Simulation** = The action is only carried out if the trigger event occurs when running a simulation. **Off** = The action is not carried out when the trigger event occurs.
Comment	Text box for comments and notes about the event.

Parts specific to I/O signal triggers

Part	Description
Active Controller	Select the system to which the I/O to use as a trigger belongs.
Signals	Displays all signals that can be used as triggers.
Trigger Condition	For digital signals, sets whether the event shall trigger when the signals are set as true or false. For analog signals, which are only available for station signals, the event shall trigger under any of the following conditions: **Greater than**, **Greater/Equal**, **Less than**, **Less/Equal**, **Equal to**, **Not equal to**.

Parts specific to I/O connection triggers

Part	Description
Add	Opens a dialog box for adding an activator signal to the Activator Signals pane.
Remove	Removes a selected activator signal.
Add >	Opens a dialog box for adding an operator symbol to the Connections pane.
Remove	Removes a selected operator symbol.
Delay (s)	Specifies the delay in seconds.

Parts specific to Collision triggers

Part	Description
Collision Type	Set the kind of collision to use as trigger. **Started** = Triggers when a collision starts. **Ended** = Triggers when a collision ends. **Near miss started** =Triggers when a near-miss starts. **Near miss ended** = Triggers when a near-miss ends.
Collision set	Select the collision set to use as trigger.

Continues on next page

The action editor parts

In the action editor you set the properties of the actions for the event. The upper part of the editor is common to all types of actions, and the lower part adjusts to the selected action.

Parts common to all actions

Part	Description
Add Action	Adds a new action that occurs when the triggering condition fulfills. You can add several different actions that either are performed at once or one at a time each time the event triggers. The following types of actions are available: **Change I/O** = Changes the value of a digital input or output signal. **Attach object** = Attaches an object to another. **Detach object** = Detaches and object from another. **Turn On/Off Timer** = Activates or deactivates the process timer. **Do Nothing** = No action occurs (might be useful for manipulating sequences of actions).
Remove Action	Removes the action selected in the Added Actions list.
Cyclic	When selected, the actions are performed one at a time each time the trigger occurs. When all actions in the list have been performed, the event will restart with the first action in the list. When cleared, all actions are performed at once every time the trigger occurs.
Added Actions	Lists all actions of the event, in the order they will be executed.
Arrow	Rearranges the order in which the actions are executed.

Parts specific to I/O Actions

Part	Description
Active Controller	Displays all systems of the station. Select the system to which the I/O to change belongs.
Signals	Displays all signals that can be set.
Action	Sets whether the event shall set the signals to true or false. If the action is connected to an *I/O Connection*, this group will not be available.

Parts specific to Attach actions

Part	Description
Attach object	Select an object in the station to attach.
Attach to	Select the object in the station to attach to.
Update position / **Keep position**	**Update position** = Moves the local origin of the attached object to the attachment point of the other object when making the attachment. For mechanisms, the attachment point is the TCP or the flange; for other objects, it is the local origin. **Keep position** = Keeps the current position of the object to attach when making the attachment.
Flange index	If the mechanism you attach the object to has several flanges (attachments points), select the one to use.
Offset Position	Optionally, specify an offset between the objects when making the attachment

Continues on next page

Part	Description
Offset Orientation	Optionally, specify an offset between the objects when making the attachment

Parts specific to Detach actions

Part	Description
Detach object	Select an object in the station to detach.
Detach from	Select the object in the station to detach from.

Parts specific to Turn On/Off Simulation Monitor actions

Part	Description
Mechanism	Selects the mechanism.
Turn Simulation Monitor On/Off	Sets whether the action shall start or stop the simulation monitor function.

Parts specific to Turn On/Off Timer actions

Part	Description
Turn On/Off Timer	Sets whether the action shall start or stop the process timer.

Parts specific to Move Mechanism to Pose actions

Part	Description
Mechanism	Selects the mechanism.
Pose	Selects between **SyncPose** and **HomePose**.
Station signal to set when Pose reached	Lists the station signals that are sent after the mechanism reaches its pose.
Add Digital	Click this button to add a digital signal to the grid.
Remove	Click this button to remove a digital signal from the grid.

Parts specific to Move Graphical Object actions

Part	Description
Graphical Object to Move	Select a graphical object in the station to move.
New Position	Sets the new position of the object.
New Orientation	Sets the new orientation of the object.

Parts specific to Show/Hide Graphical Object actions

Part	Description
Graphical Object	Select a graphical object in the station.
Show/Hide	Sets whether the object is shown or hidden.

Parts specific for a Call VSTA Macro action

Part	Description
Available Macros	Displays all available macros in the station.

10.5. Activate Mechanical Units

To activate or deactivate mechanical units manually

1. Click **Activate Mechanical Units** to bring up a dialog box.
2. In the **Activate Mechanical Units** dialog box, select the check boxes for the mechanical units to set as active. When activating a mechanical unit that shares a common drive unit, the other mechanical unit sharing that drive unit will be deactivated automatically.

10.6. Simulation Control

Running a simulation

1. Optionally, click **Enable Process Timer**.
2. In the **Simulation Control** group, click the **Play** button.

   The output window will now display messages from the virtual controller.
3. To pause the simulation, click the **Pause** button.
4. To stop the simulation, click the **Stop** button.
5. To reset the simulation to its initial state, click the **Reset** button.

10.7. I/O Simulator

Setting I/O signals using the I/O Simulator

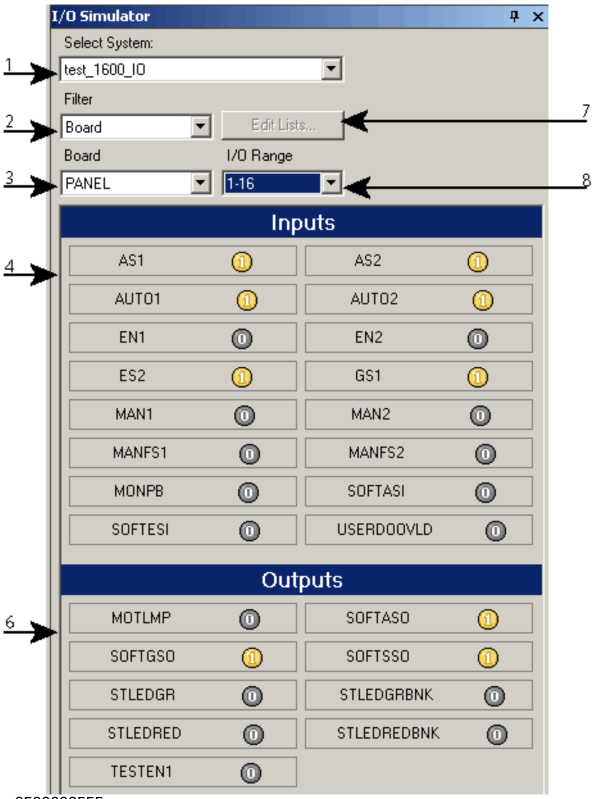
1. Click **I/O Simulator**. This opens the I/O simulator.
2. If the station contains several systems, select the appropriate one in the **Select System** list.
3. In the **Filter** list and **I/O Range** list, make selections that display the signals to set. Depending on the filter used, you might also set a filter specification.
4. To change the value of a digital I/O signal, click it.

   To change the value of an analog signal, type the new value in the value box.

The I/O Simulator window

With RobotStudio's I/O simulator you view and manually set existing signals, groups and cross-connections during program execution, thus making it possible to simulate or manipulate the signals.

The I/O simulator displays the signals for one system at a time in groups of 16 signals. For handling large sets of signals, you can filter which signals to display and also create custom lists with favorite signals for quick access.



xx0500002555

Part	Description
1.	**Select System**. Select the system whose signals you want to view.
2.	**Filter type**. Select the type of filter to use.
3.	**Filter Specification**. Select the filter for limiting the signal display. For example, if Board is set as filter type, then you select the board whose signals you want to view.

Continues on next page

Part	Description
4.	**Inputs**. Displays all input signals that pass the applied filter. If more than 16 signals pass, only 16 signals at a time are displayed. Then use the **I/O range** list to select the signals to view.
5.	**Outputs** Displays all output signals that pass the applied filter. If more than 16 signals pass, only 16 signals at a time are displayed. Then use the **I/O range** list to select the signals to view.
6.	**Edit Lists**. Click this button to create or edit lists of favorite signals.
7.	**I/O Range**. When more than 16 signals pass the filter, use this list to select the range of signals to display.

Types of signal filters

Filter	Description
Board	Displays all signals on a specific board. To select a board, use the **Filter Specification** list.
Group	Displays all signals that belong to a specific group. To select a group, use the **Filter Specification** list.
User List	Displays all signals in a favorite list. To select a list, use the **Filter Specification** list.
Digital Inputs	Displays all digital input signals of the system.
Digital Outputs	Displays all digital output signals of the system.
Analog Inputs	Displays all analog input signals of the system.
Analog Outputs	Displays all analog output signals of the system.

Signal icons

 xx0500002558	Digital signal with value 1.
 xx0500002558	Digital signal with value 0.
 xx0500002558	The cross in the upper right corner indicates that the signals are a cross-connection.
 xx0500002558	The -1 in the upper right corner indicates that the signal is inverted.
 xx0500002562	Value box for groups or analog signals.

10.8. Monitor

The TCP Trace tab

Enable TCP Trace	Select this check box to activate tracing of the TCP path for the selected robot.
Trace length	Specify the maximum length of the trace in millimeters.
Trace Color	Displays the color of the trace when no alerts are activated. To change the color of the trace, click the colored box.
Alert color	Displays the color of the trace when any of the alerts defined on the **Alerts** tab exceeds a threshold value. To change the color of the trace, click the colored box.
Clear Trace	Click this button to remove the current trace from the graphics window.

The Alerts tab

Enable Simulation Alerts	Select this check box to activate simulation alerts for the selected robot.
Log Alerts to Output Window	Select this check box to see a warning message when a threshold value is exceeded. If TCP trace is not enabled, this is the only display of the alert.
TCP Speed (m/s)	Specify the threshold value for TCP speed alerts.
TCP Acceleration (m/s2)	Specify the threshold value for TCP acceleration alerts.
Wrist Singularity (deg)	Specify how close joint five can be to zero rotation before alerting.
Joint Limits (deg)	Specify how close each joint can be to its limits before alerting.

10.9. Record Movie

Prerequisites

For optimal results, first configure the options, see *Options:General:Screen Recorder on page 179*.

Recording the screen

1. In the **Record Movie** group, click **Record application** to capture the entire application window, or **Record graphics** to capture just the graphics window.
2. When you are done, click **Stop Recording**. A dialog box appears in which you may choose to save the recording or discard it.
3. Click **View Recording** to playback the latest capture.

Recording the simulation

1. In the **Record Movie** group, click **Record Simulation** to record the next simulation to a video clip.
2. When you are done, click **Stop Recording**.

   The simulation is saved in a default location which is displayed in the output window.
3. Click **View Recording** to playback the recording.

   The recording of simulation starts when you click **Play** in the **Simulation** tab.

NOTE!

Record Simulation gives better output quality than **Record application** or **Record graphics**.

10.10 Conveyor Tracking Mechanism

10.10.1. Conveyor Tracking

Overview

Conveyor tracking is the function where the robot follows a workobject mounted on a moving conveyor.

This section describes *how to create a conveyor*, *add and remove objects to and from the conveyor, create targets during tracking*, and *simulate conveyor.*

For more information, see the *Application manual - Conveyor tracking*.

Conveyor tracking mechanism

This procedure describes the workflow for making a conveyor tracking system work in RobotStudio.

1. Create a conveyor mechanism. See *Create Conveyor mechanism on page 247*.
2. Setup the conveyor. See *Setting up a conveyor on page 189*and *Encoder Unit on page 296*.

   NOTE! To setup a conveyor tracking station with two robots working on the sameconveyor. See *Setting up a conveyor tracking station with two robots working on the same conveyor on page 56*.
3. Jog the conveyor as well as the robot and teach some targets. See *Mechanism Joint Jog on page 340*.
4. Simulate the conveyor. See *Conveyor Simulation on page 271*.
5. Remove objects from conveyor. See *Removing objects from conveyor on page 190*.

10.10.2. Conveyor Simulation

Running a conveyor simulation

1. Create **Action Instructions**. See *Action Instruction on page 215*.

   NOTE! Create the following five action instructions along with Move instructions: `ConfL\Off,ActUnit CNV1,WaitWObj Workobject_1, DropWObjWorkobject_1` and `DeactUnit CNV1`.

   The following program is an example showing how the sequence of instructions appears:

```
ConfL\Off;
MoveJ p0, vmax, fine, tool1;
ActUnit CNV1;
WaitWObj wobjcnv1;
MoveL p10, v1000, z1, tool1\Wobj:=wobjcnv1;
MoveL p20, v1000, z1, tool1\Wobj:=wobjcnv1;
MoveL p30, v500, z20, tool1\Wobj:=wobjcnv1;
MoveL p40, v500, fine, tool1;
DropWObj wobjcnv1;
MoveL p0, v500, fine;
DeactUnit CNV1;
```

   en080000451

   NOTE! If an error occurs while executing the program, the controller reaches the Guard state. In this state, RobotStudio cannot execute the program during the next simulation. To recover from this state, open the **Control Panel** and switch to **Manual Mode** and then to **Auto Mode**.

   For more information, see *Application manual - Conveyor Tracking*.

2. Synchronize to VC. See *Synchronize to VC on page 275*.
3. Set up the Simulation. See *Simulation Setup on page 257*.
4. Click **Simulation**.

   The Conveyor Simulation dialog appears.

5. In the **Conveyor Speed** box, set the speed during simulation.

   NOTE! To move the conveyor in the backward direction, select the **Reverse** check box.

6. Click **Apply**.
7. Click **Play** to run the simulation.

   NOTE! The conveyor speed and direction can be changed while running the simulation.

NOTE!

To jump the conveyor back to the start position, click **Reset**. This button remains enabled as long as the station has at least one conveyor.

11 The Offline Tab

11.1. Overview

The Offline tab

The Offline tab contains the controls for synchronization, configuration and tasks assigned to the VC.

11.2. Synchronize to Station

Synchronizing to the station

1. Click **Synchronize to Station** to bring up a dialog box.
2. Select the paths to be synchronized to the station from the list.
3. Click **OK**.

11.3. Synchronize to VC

Synchronizing to the virtual controller

1. In the **Offline** browser and the Active task list, select the task you want to synchronize.
2. Click **Synchronize to VC** to bring up a dialog box.
3. Select the check boxes for the elements to synchronize.

Element	In this column, all elements that can be synchronized, such as data, paths and targets, are displayed. To include the element in the synchronization, select the check box in front of the element name.
Module	In this column, the module in the RAPID program to which the element will be synchronized is displayed. To change the module for an element, right-click the current one and select, or type a new in the list.
Storage class	In this column, the storage class of each element is displayed. For detailed information about storage classes for the different elements, see the *RAPID Reference Manual*. To change the storage class for an element, right-click the current one and select, or type a new in the list. The column is visible only in advanced mode.
Inline	In this column, you can specify whether the target for the move instruction shall be declared in the data section (*False*) or written directly in the instruction (*True*). The column is visible only in advanced mode.
Select path targets	Click this button to select in the Elements column all targets in the path. This button is available only in advanced mode after a path is selected.
Simple/Advanced	Click this button to toggle between the simple and advanced display modes. In simple mode, the tooldata, workobjects, storage classes and inline columns will not be displayed.

4. Optionally, click **Advanced** and review or change the settings for the synchronization.
5. Click **OK**.

11.4 Backup

11.4.1. Backing up a system

Overview

When backing up a system you copy all the data needed to restore the system to its current state:

- Information about software and options installed on the system.
- System's home directory and all its content.
- All robot programs and modules in the system.
- All configuration and calibration data of the system.

Prerequisites

To backup a system you must:

- Write access to the controller
- Logged on to the controller with appropriate grants. For more information, see *User Authorization on page 137*.

Creating a Backup

To create a backup, follow these steps:

1. In the **Offline** browser, select the system you want to backup.
2. Click **Backup** and select **Create Backup**.

   The Create Backup dialog box appears.
3. Enter a new backup name and specify a location for the backup, or keep the default ones.
4. Click **Backup**.

   The progress of the backup appears in the Output window.

Result

When the back up is complete you will have a folder with the name of the back up in the specified location. This folder contains a set of subfolders which all together comprises the backup:

Folder	Description
Backinfo	Contains information necessary for re-creating the system's software and options from the mediapool.
Home	Contains a copy of the system's home directory content.
Rapid	Contains one subfolder for each task in the system's program memory. Each of these task folders contains separate folders for program modules and system modules.
Syspar	Contains the system's configuration files.

CAUTION!

If the content of the Backup folder is changed, it is not possible to restore the system from backup.

11.4.2. Restoring a system from backup

Overview

When restoring a system from backup, the current system gets the same content as when the backup was performed. Restoring a system replaces the following contents in the current system with the content from the backup:

- All RAPID programs and modules in the system.
- All configuration and calibration data of the system.

NOTE! The system's home directory and all its content are copied from the backup to the current system.

Prerequisites

To restore a system you must:

- Write access to the controller.
- Logged on to the controller with appropriate grants. For more information, see *User Authorization on page 137*.

Restoring a system

To restore a system, follow these steps:

1. In the **Offline** browser, select the system you want to restore.
2. Click **Backup** and select **Restore Backup**.

   The Restore from Backup dialog box appears.
3. In the **Restore from Backup** dialog box, select which backup to use for restoring the system.
4. Click **Restore**.

   The progress of the restore appears in the Output window.
5. When the restore is complete, restart the controller to load the restored system. See *Restarting a VC on page 66*.

NOTE!

Before proceeding, make sure that the system from the backup is compatible with the controller you are restoring.

System ID and Template ID

If the system from the backup does not originate from the controller you are restoring, you will get a message about the mismatch.

The following table describes the possible cases:

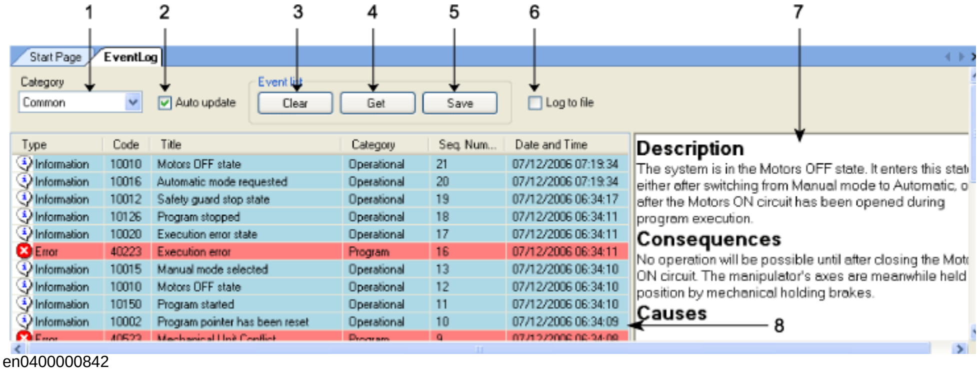
Case	Description
System ID and Template ID are identical	When both System ID and Template ID are identical between current system and backup system, there is no difference between the systems.

Continues on next page

Case	Description
Mismatch in System ID	If there is a mismatch in System ID there could be a minor difference between current system and backup system, as for example different languages. RobotStudio warns regarding mismatch. **NOTE!** Before you restore, make sure that the backup system is compatible with the current system.
Mismatch in both System ID and Template ID	There can be mismatch in both System ID and Template ID between current system and backup system, as for example different robot types. RobotStudio warns that this is not a recommended action. **NOTE!** You must have a correct backup to the system that you restore.

11.5. Events

The Evet Log tab



en0400000842

Parts

1. **Category**

   This list contains the different event categories. The default category *Common* includes all categories.

2. **Auto Update**

   This checkbox is checked by default, meaning that new events will appear in the list when they occur. Clearing the checkbox disables this automatic update. Checking it again, however, will fetch and display the events missed while it was cleared.

3. **Clear**

   This button clears the current event record. This does not affect the event log of the controller, which can be retrieved again by clicking the **Get** button.

4. **Get**

   This button retrieves and displays all events currently stored in the controller.

5. **Save**

   This button saves the event records of the selected event categories to log files on the computer.

6. **Log to file**

   This checkbox enables all events currently shown by the Event Log to be saved to a log file on the computer. If it remains checked the log file will be updated with new events as they occur.

7. **Event Information**

   This box displays information about the event selected in the event list.

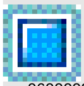
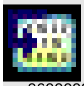
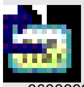
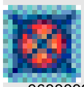
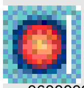
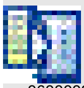
8. **Event Record**

   The event record is shown as a list of events of a selected category. The severity of the event is indicated by color: blue for information, yellow for warning and red for an error which needs to be corrected in order to proceed.

11.6. RAPID editor

Buttons on the RAPID editor toolbar

The table below describes the buttons on the **RAPID editor** toolbar:

Button	Description
 xx0600003383	**Start** Starts the execution of all normal RAPID tasks in the system.
 xx0600003384	**Stop** Stops the execution of all normal RAPID tasks in the system.
 xx0600003385	**Step over** Starts and executes one statement in all normal tasks in the system.
 xx0600003386	**Step in** Starts and executes into a routine, while stopping at the beginning of the routine.
 xx0600003387	**Step out** Executes all remaining statements of the current routine, and stops after the call to the current routine.
 xx0600003394	**Ignore breakpoints** Ignores all breakpoints during simulation.
 xx0600003388	**Toggle breakpoint** Toggles a breakpoint at the cursor.
 xx0600003389	**Apply changes** Applies the changes to the module in the program editor to the system.
 xx0600003390	**Check program** Verifies the syntactic and semantic correctness of all modules in the system.
 xx0600003391	**Print** Prints the contents of the program editor.

Continues on next page

About RAPID Editor

The RAPID editor enables you to view and edit programs loaded into the controller program memory. With the RAPID editor you edit the RAPID code of the program's modules. Each module you open will appear in a program editor window of its own, where you type in the code. For more information, see *Using the RAPID editor on page 113*.

RAPID Editor functions

RAPID editor has the following functions:

- Cut, copy, paste and drag and drop - supports clipboard handling of rich text.
- Undo and redo - supports undo and redo operations.
- Go To Line - navigates to a specific line within the Editor using a keyboard shortcut.
- Selection modes - possibility to select text by character, row and column.
- Line numbers - appears in the left margin of the editor.
- Keyboard shortcuts - see *Keyboard Shortcuts on page 49*.

Programming functions

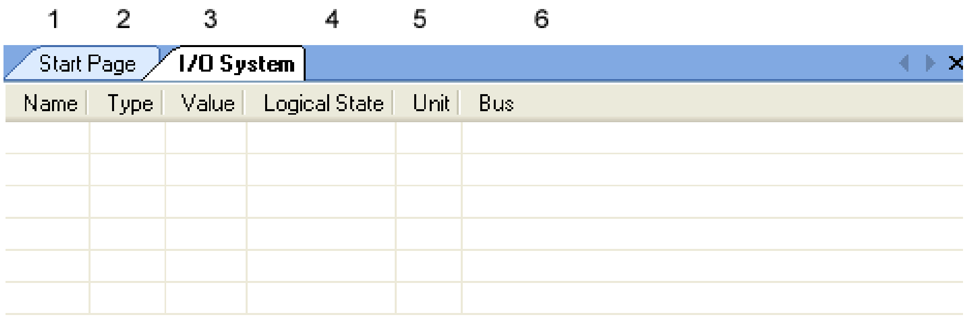
The RAPID Editor has the following RAPID specific functions.

Function	Description
Syntax coloring	Writes each part of the code with a specific color, based on its syntactic function in RAPID. Helps to quickly recognize words and misspellings. The automatic coloring of the text is based on the syntax of the RAPID language.
Parameter Info	Shows available parameters in a tool tip for the RAPID instructions you type. Helps to type in a procedure or function call in the editor by showing all the optional and required arguments that must be specified.
Pick list	Helps to navigate through a hierarchy of small code snippets which can be inserted at the insertion position.
Routine list	Is a combo box containing all routines in the RAPID module of the editor.
Complete word	Automatically completes the keyword by predicting the keyword or identifier that you want to type in without you actually typing.

11.7. Inputs / Outputs

The I/O system window

From the I/O system you can view and set input and output signals.



en0400000840

Parts

1. The **Name** column

   This column shows the name of the signal. The name is set by the I/O unit's configuration and cannot be changed from the I/O system.

2. The **Type** column

   This column shows which type of signal it is, by using any of the abbreviations described below. The signal type is set by the I/O unit's configuration and cannot be changed from the I/O system.

Abbreviation	Description
DI	Digital input signal
DO	Digital output signal
AI	Analog input signal
AO	Analog output signal
GI	Group of signals, working as one input signal
GO	Group of signals, working as one output signal

3. The **Value** column

   This column shows the value of the signal. The value can be changed by double-clicking the signal row.

4. The **Logical State** column

   This column shows whether the signal is simulated or not. When a signal is simulated, you specify a value that overrides the actual signal. Changing the logical state by turning the simulation on or off can be done from the I/O system.

5. The **Unit** column

   This column shows to which I/O unit the signal belongs. This is set by the I/O unit's configuration and cannot be changed from the I/O system.

6. The **Bus** column

   This column shows to which I/O bus the signal belongs. This is set by the I/O bus' configuration and cannot be changed from the I/O system.

11.8. System Builder

Functions of the System Builder

For procedures using the various functions of the System Builder, see *System Builder Overview on page 139*.

11.9. Rapid Tasks

Prerequisites

You must have write access to the controller to be able to open the Task Window. You must also be logged on to the controller as a user who has either of the grants *Full access* or *Execute program*.

Different states

The following table shows different states when actions are not possible in the Task Window:

If...	RobotStudio gives a message that informs the user that...
the user does **not** have the grant *Execute program* or *Full access*	the operation is not possible.
the user changes from manual mode to automatic mode, or vice versa, the user loses the write access and	the operation is not possible.
the motors are in off state	the Start operation is not possible.

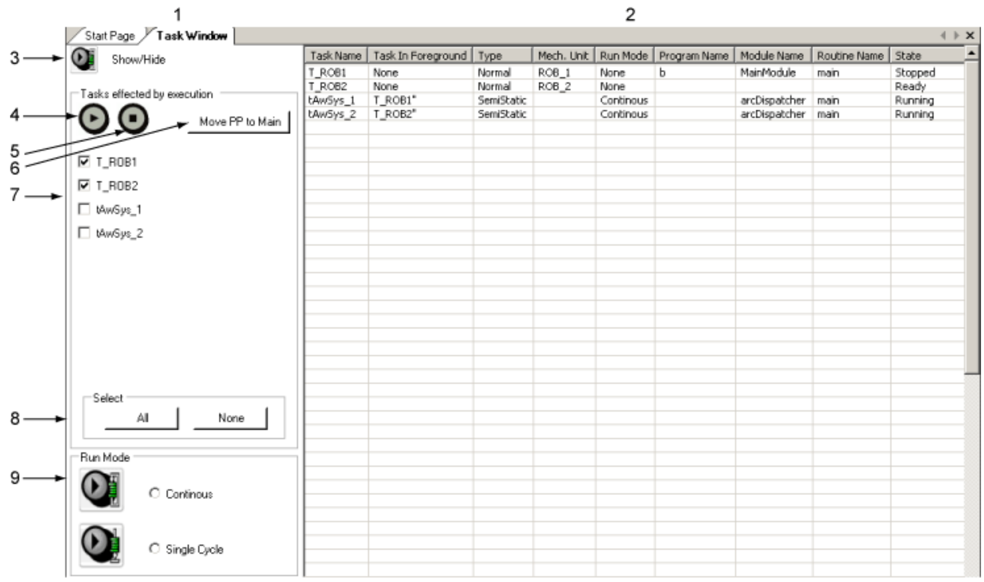
The safety system of the controller

It is not possible to override the controller’s safety system, that is, you cannot stop a background task (Static and SemiStatic) that has the TrustLevel set to another value than NoSafety.

For detailed information about the different TrustLevel values, see *Technical reference manual - system parameters*.

The Task window

The Task window is divided in two parts, one left part with functionality (1) to operate the task(s) and one right part (2) with the task list and information about the tasks.



en0500002159

Continues on next page

If the prerequisites are met, you can operate the task, such as start and stop the task, move the program pointer to main and set the run mode. This part will be opened as default when you open the Task Window, but you can hide this part.

The following table describes the different buttons in the functionality part:

3	The **show/hide** button. Shows/hides the functionality part.
4	The **start** button. Starts the selected tasks. **Note!** You can start Normal tasks, but you can only start a Static or SemiStatic task if the TrustLevel is set to NoSafety.  - **Danger!** When starting a task, the manipulator axes may move very quickly and sometimes in unexpected ways! Make sure no personnel is near the manipulator arm!
5	The **stop** button. Stops the selected tasks. **Note!** You can stop Normal tasks, but you can only stop a Static or SemiStatic task if the TrustLevel is set to NoSafety.
6	The **Move PP to Main** button. Moves the program pointer to Main. Applies to all selected tasks.
7	The **Task names** check box. Selected check box indicates that the task will be operated.
8	The **Select** buttons. Select All or None of the tasks.
9	The **Run Mode** buttons. Set run mode to Continuous or Single cycle. The **Run Mode** indicates the mode of the controller.

Task Name	The task name.
Task in Foreground	Which task that is in the foreground, for example main. Gives indirectly the priority.
Type	Normal/Static/SemiStatic: • Normal: The task reacts on START/STOP requests. The task is stopped when an emergency stop occurs. The TrustLevel for the Normal task corresponds to the TrustLevel NoSafety. • Static: At warm start of the controller, the task restarts at the current position. • SemiStatic: The task restarts from the beginning at all warm starts of the controller. Default value is SemiStatic. **Note!** If the type of task is Static or SemiStatic, the following is valid only for the Task Window : If the TrustLevel is set to NoSafety, the task can be stopped by the stop button on the Task Window. If the TrustLevel is set to SysFail, SysHalt or SysStop, the task *cannot* be stopped.
Mech. Unit	If the task has robot specific instructions, this specifies which robots they are valid for.
Run Mode	Continuous or single cycle.
Program Name	The name of the program in the specific task.
Module Name	The current module name.
Routine Name	The current routine name.

Continues on next page

State	Ready/Running/Stopped: • Ready: The program has no PP (program pointer). To get a PP to the program, click the **Move PP to main** button. • Running: The program is running. • Stopped: The program has stopped.

You can operate the task list as following:

- All columns can be adjusted in the standard windows way - drag or double-click between the column headers.
- If all the columns do not fit into the window, a horizontal scroll bar will occur.
- If all the tasks do not fit into the window, a vertical scroll bar will occur.
- You can sort the task information in all columns by clicking the column heading.

11.10. Restart

Restarting a VC

1. In the **Layout** browser, select the controller to restart.
2. Click **Restart** and then click either **Warmstart** to activate the changes and restart the VC or **I-start** to restart the VC with the current system and default settings.

11.11. Shutdown

Shutting down a VC

1. In the **Offline** browser, select the controller to shut down.
2. Click **Shutdown** and then click either **Shutdown** to close the connection to the VC or **Terminate Virtual Controllers** to end all VC processes.

11.12. New Module

Creating a module

1. Click **New Module** to bring up a dialog box.
2. Select the module type and permission and click **Create**.

11.13. Load Module

Loading a module

1. Click **Load Module** to bring up a dialog.
2. Browse to the location of the module to be loaded to your station and click **Open**.

11.14. Load Program

Loading a program

1. Click **Load Program** to bring up a dialog box.
2. Browse to the location of the program to be loaded to your station and click **Open**.

11.15. Add signals

Parts

Signal Name Expression	Defines the name for one or more signals.
Signal Type	Defines the type of signal.
Assigned Unit	Defines the I/O unit to which the signal belongs.
Identification Label	Optionally, offers filtering and sorting based on this category.
Unit Mapping Expression	Defines the bits in the I/O memory map of the assigned unit to which the signal is mapped.
Signal Category	Optionally, offer filtering and sorting based on this category.
Invent Physical Value	Applies an inversion between the physical value of the signal and its logical representation in the system.

NOTE!

You must have write access to the controller to be able to open the add signal window.

11.16. Set Task Frames

Modifying Task frame

1. In the **Offline** browser, click **Set Task Frame.**
   The **Modify Task Frames** dialog box appears.
2. Set the reference to **World**, **UCS,** or **Local**.
3. Edit the position and orientation of task frames in the **Task Frames** coordinate box.
   This updates the position of all frames relative to the system.
4. Click **Apply.**

11.17. System Configuration

Overview

The system configuration window contains functions for making and viewing advanced system configurations, such as changing controller and baseframe positions, calibrating and setting up external axes.

The left side of the system configuration window contains a hierarchical tree with which you browse to different aspects of the system. The right side contains a property sheet for the aspect selected in the tree. Below are short descriptions of the property sheets for each aspect node of the system configuration.

CAUTION!

Changing the system configurations may result in corrupted systems or unexpected robot behaviors. Be sure to understand the effects of the changes before proceeding.

The System node

The system node contains a box with information about the system and a button for loading new parameters (configuration files) to the system.

The task node

The task node has no property page.

The mechanism folder node

The property page of this node contains controls for mapping and setting axis and joints. It is from this page you set up external axes.

The mechanism library node

The property page of this node contains controls for changing the baseframe of the robot or mechanism. Here, too, you specify whether the baseframe is moved by another mechanism (coordinated motion), like a track external axis.

Updating the baseframe position

1. Move the mechanical unit (robot or external axis) to its new location using the ordinary tools for moving and placing objects.
2. In the **Offline** browser, select the controller for the mechanical unit.
3. Click **System Configuration** to bring up a dialog box.

   NOTE! The System configuration dialog box can also be opened by:

   - In the **Paths&Targets** browser, right-click a station.
   - Select **Configuration** and click **System Configuration**.
4. Select the node for the mechanical unit in the hierarchical tree. The baseframe property sheet for the robot is now displayed.

Continues on next page

5. Select the baseframe position values to use after restarting the robot.

Select	To
Controller values	Reset all changes to the baseframe made since the last time the system was started.
Stored station values	Reset all changes made to the baseframe since the last time the station was saved. Optionally, you can enter new values in the baseframe coordinate boxes (relative to the controller world coordinate system).
Use current station values	Read and use the current location of the baseframe. Optionally, you can enter new values in the baseframe coordinate boxes (relative to the controller world coordinate system).

6. Click **OK**.

11.18. Encoder Unit

Configuring a Conveyor Encoder Unit

1. Click **Encoder Unit**.

   The Configure Conveyor Encoder Unit dialog box appears.

   NOTE! The Configure Conveyor Encoder Unit dialog box can also be opened by:

   - In the **Paths&Targets** browser, right-click a station.
   - Select **Configuration** and click **Encoder Unit**.

2. Select **CNV1** from the **Mechanical Unit** list.
3. In the **Parameters** box, enter the values for **Maximum Distance, Minimum Distance, Queue Tracking Distance** and **Start Window Width**.

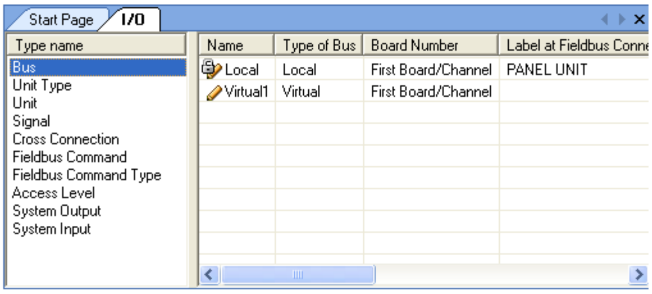
   NOTE!

   If any of the parameter values are changed, the controller must be restarted.

4. Click **OK**.
5. Click **Yes** to restart the controller.

11.19. Configuration editor

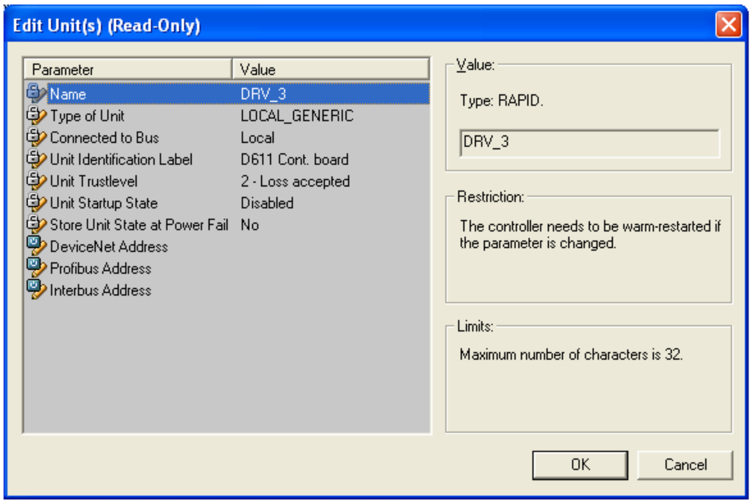
Configuration editor

1  2

en0400000838

1. The **Type name** list displays all available configuration types for the selected topic. The list of types is static. This means you cannot add, delete or rename types.
2. The **Instance** list displays all system parameters of the type selected in the **Type name** list. Each row in the list is an instance of the system parameter type. The columns show each specific parameter and its value for each instance of the parameter type.

Instance editor

1  2 3 4

en0400000839

1. The **Parameter** list displays the parameters and their value for the open instance.
2. The **Value** box displays the type of parameter and the value of the parameter.
3. The **Restriction** box displays the restrictions for the parameter. The restrictions hasve to be fulfilled to update the controller database.
4. The **Limits** box displays the limits for the parameter.

Icon	Description
 xx	Editable parameter

Continues on next page

Continued

Icon	Description
 xx	Parameter that is not applicable for this instance and therefore not editable.
 xx	Parameter that is read-only.

11.20. Load Parameters

Prerequisite

You must have write access to the controller.

Loading parameters

1. In the **Offline/Online** browser, select the configuration node.
2. Click **Load Parameters** to bring up a dialog box.
3. In the dialog box, select how you want to combine the parameters in the configuration file to load with the existing parameters:

If you want to...	**then select...**
replace the entire configuration of the topic with the one in the configuration file.	**Delete existing parameters before loading**
add new parameters from the configuration file to the topic, without modifying the existing ones.	**Load parameters if no duplicates**
add new parameters from the configuration file to the topic and update the existing ones with values from the configuration file. Parameters that only exist in the controller and not in the configuration file will not be changed at all.	**Load parameters and replace duplicates**

4. Click **Open** and browse to the configuration file to load. Then click **Open** again.
5. In the information box, click **OK** to confirm that you want to load the parameters from the configuration file.
6. When the loading of the configuration file is finished, close the Select mode dialog box.

   If a restart of the controller is necessary for the new parameters to take affect, you will be notified of this.

11.21. Save System Parameters

Overview

The system parameters of a configuration topic can be saved to a configuration file, stored on the PC or any of its network drives.

The configuration files can then be loaded into a controller. They are thereby useful as backups, or for transferring configurations from one controller to another.

File-naming conventions

The configuration files should be named with a name that relates to their corresponding topics. When saving configuration files, the correct name for each file will be suggested by default.

Saving one configuration file

1. In the **Offline/Online** browser, expand the **Configuration** node and select the topic to save to a file.
2. Click **Save System Parameters**.
3. In the **Save As** dialog box, browse for the folder to save the file in.
4. Click **Save**.

Saving several configuration files

1. In the **Robot View Explorer**, select the **Configuration** node.
2. Click **Save System Parameters**.
3. In the **Save System Parameters** dialog box, select the topics to save to files. Then click **Save**.
4. In the **Browse for Folder** dialog box, browse for the folder to save the files in.

   Then click **OK**.

   The selected topics will now be saved as configuration files with default names in the specified folder.

12 The Online Tab

12.1. Overview

The Online tab

The Online tab contains controls for managing the real controller.

12.2. Add Controller

Adding a controller

1. Click **Add Controller** to bring up a dialog box in which all available controllers are listed.
2. If the controller is not found in the list, type its IP address in the **IP Address** box, and then click **Refresh**.
3. Select the controller in the list and click **OK**.

12.3. Request Write Access

Overview

You can access the controllers in your Robot View with either read only access or Write access. You need Write access for editing programs and configurations or in any other way change data on the controller.

Prerequisites for Write access

You can get Write access to any controller as long as the prerequisites are fulfilled.

When the Controller is in Mode:	This has to be fulfilled:
Auto	The Write access must not be taken by any other user.
Manual	The remote Write access must be granted on the FlexPendant. For safety reasons, a FlexPendant user can also recall this remote Write access in manual mode.

If the prerequisites are not fulfilled you will be denied, or lose, the Write access. This means that if you have Write access in auto mode and the controller is switched over to manual mode you will lose the Write access without any warning. This is because the FlexPendant unit by default has the Write access in manual mode, for safety reasons. The same will happen if the remote Write access in manual mode is recalled from the FlexPendant unit.

Result

The Controller Status window will be updated when the request for Write access is granted.

If the Write access is denied, a message is displayed.

12.4. Release Write Access

Overview

You can access the controllers in your Robot View with either read only access or write access. Since several users can be logged on to one controller but only one can have write access, you can release the write access when you do not need it anymore.

Result

The Controller Status window will be updated when your access right has changed from read/write to read only.

12.5. Import Options

Importing system options

1. Click **Import System Options** to bring up a dialog box.
2. In the **Option source** box, enter the path to the folder where the options to import are located. You can also click the Browse button and browse to the folder.
3. In the **Media pool destination** box, enter the path to the media pool you want to store the options in. You can also click the Browse button and browse to the media pool folder.
4. Select the options to import and click **Import**.

   To select several options at once, do one of the following:

To select	then hold down
several adjacent options	the SHIFT key and select the first and the last option.
several non-adjacent options	the CTRL key and select each option.

5. Click **OK**.

Removing system options

1. Click **Import System Options** to bring up a dialog box.
2. In the **Media pool destination** list, enter the path to the media pool from which you want to delete the options. You can also click the Browse button and browse to the media pool folder.
3. Select the options to delete and click **Remove**.

   To select several options at once, do one of the following:

To select	then hold down
several adjacent options	the SHIFT key and select the first and the last option.
several non-adjacent options	the CTRL key and select each option.

4. Click **OK**.

12.6. User Accounts

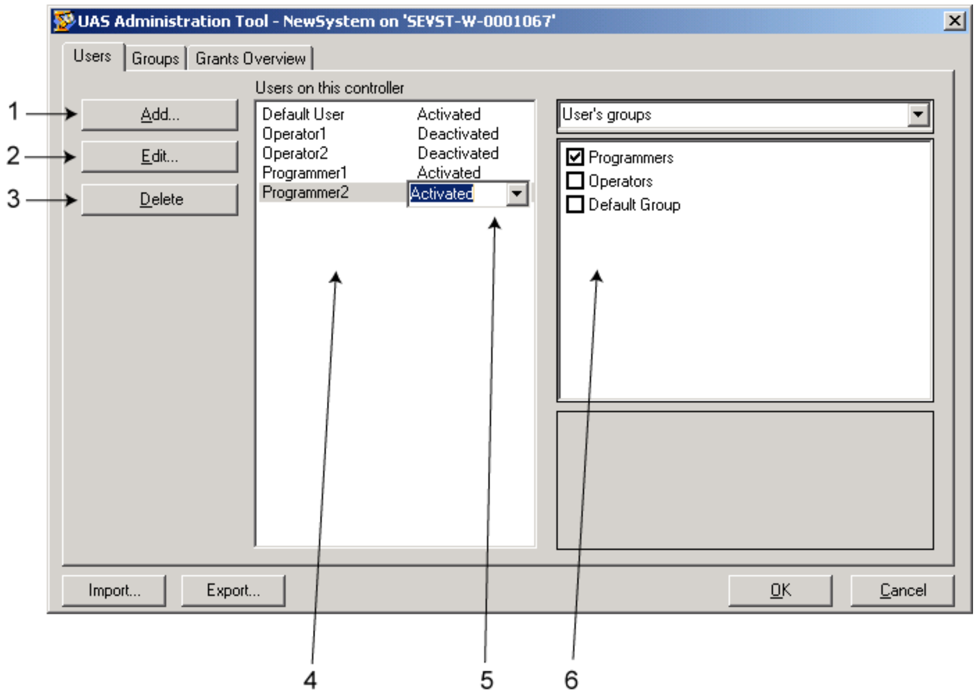
Overview

All the procedures below require the following steps to be taken before managing the details:

1. In the **Online** browser, select the controller to which you want to manage a user. or group
2. From the **Online** tab, request write access to the controller.
3. On the **Online** tab, click **User Accounts**.

Users tab

With the Users tab you set which users will be able to log on to the controller and which groups the users shall belong to.



en0400001104

Users tab Parts

1. The **Add** button. Opens a dialog box for adding new users.
2. The **Edit** button. Opens a dialog box for changing the user's log on name and password.
3. The **Delete** button. Deletes the selected user account from the controller.
4. The **Users on this Controller** list. Shows the user accounts defined on this controller. The list has two columns:

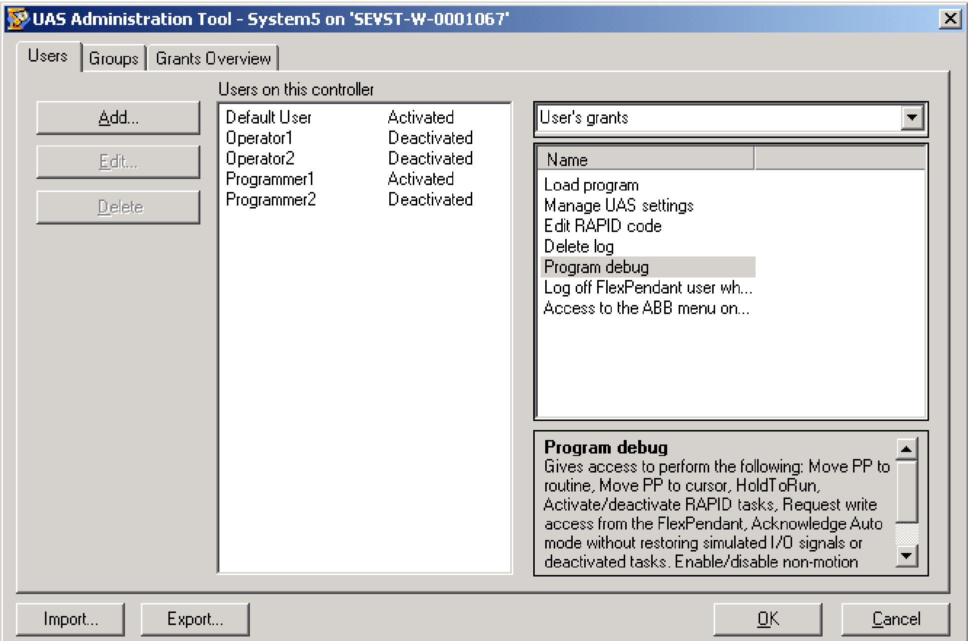
Column	Description
User	The name of the user account
Status	Shows if the account is activated or deactivated. When deactivated, it is not possible to log on using that account.

Continues on next page

5. The **Activated/Deactivated** item box. Changes the status of the user account.
6. The **User's groups/User's grants** list.

   The **User's groups** list shows which group(s) the user is a member of. For changing the membership of a group, select or clear the checkbox in front of the group name.

   The **User's grants** list shows the available grants for the selected User's group(s). When selecting a grant from the User's grants list, a description of the selected grant appears.



en0500001572

Adding a user

1. On the **Users** tab, click **Add** to bring up a dialog box.
2. In the **User Name** box, enter the user name. Use only characters from the ISO Latin -1 character set and no more than 16 characters.
3. In the **Password** box, enter the user's password. The password you type in will not be visible. Use only characters from the ISO Latin -1 character set and no more than 16 characters.
4. In the **Retype Password** box, enter the user's password again.
5. Click **OK** to add the new user and close the dialog box.
6. Click **OK**.

Deleting a user

1. On the **Users** tab, select the user to delete from the **Users on this controller** list and click **Delete**.
2. To the question **Are you sure you want to remove this user?**, answer **Yes**.
3. Click **OK**.

Continues on next page

Setting up group membership

1. On the **Users** tab, select the user from the **Users on this controller** list.
2. In the **User's groups** list, select the groups the user shall be a member of.
3. Click **OK**.

Changing a name or password

1. On the **Users** tab, select the user to edit from the **Users on this controller** list and click **Edit user**.

   This opens the **Edit** dialog box.
2. To change the user name, enter the new name in the **User Name** box. Use only characters from the ISO Latin -1 character set and no more then 16 characters.
3. To change the password, enter the new password in the **Password** box, then retype the password in the **Retype Password** box. Use only characters from the ISO Latin -1 character set and no more then 16 characters.
4. Click **OK** to save the changes to the user and close the dialog box.
5. Click **OK**.

Activating or deactivating a user

1. On the **Users** tab, select the user from the **Users on this controller** list and click the status text (Activated or Deactivated). An item box appears and you can change the status.

   The user's new state is now displayed in the status column of the **Users on this controller** list.
2. Click **OK**.

Continues on next page

Exporting a user list

1. On the **Users** tab, select the user from the **Users for this Controller** list and click **Export**.

   This opens a **Save as** dialog box, in which you specify the name and location for the file with the user list.

Importing a user list

1. On the **Users** tab, select the user from the **Users for this Controller** list and click **Import**.

   This opens an **Open file** dialog box, in which you browse to the file with the list to import.

   When you have selected the file, the ImportOptionsForm dialog appears.

Select ...	Description
Delete existing users and groups before importing	Earlier groups and users will be deleted.
Advanced options	A new dialog appears. Import users but don't replace duplicates means that you will not replace the existing users. Import users and replace duplicates means that you will replace the existing users. Import groups but don't replace duplicates means that you will not replace the existing groups. Import groups and replace duplicates means that you will replace the existing groups.

Adding a group

1. On the **Groups** tab, click **Add**.

   This opens the **Add new group** dialog box.
2. In the **Group Name** box, enter the name of the group. Use only characters from the ISO Latin -1 character set and no more than 16 characters.
3. Click **OK** to add the new group and close the dialog box.
4. Click **OK**.

Renaming a group

1. On the **Groups** tab, select the group to rename from the **Groups on this controller** list and click **Rename**.

   This opens the **Rename Group** dialog box.
2. In the **Group Name** box, enter the name of the group. Use only characters from the ISO Latin -1 character set and no more than 16 characters.
3. Click **OK** to rename the group and close the dialog box.

Continues on next page

4. You will now be asked if you want the users who belong to this group to continue belonging to a group with the old name as well as the new one.

Click	To
Yes	Let the users of the group be members to groups with both the old and the new name. However, the old group will no longer be defined in the controller's UAS since it is replaced by the new group. This option might be useful if you plan to recreate the old group, or copy the user's settings to another controller who has the old group defined.
No	Delete the user's memberships to the old group. This is to just replace the old group name with the new one.
Cancel	To cancel the change and keep the old group name, with its user's memberships.

5. Click **OK**.

Deleting a group

1. On the **Groups** tab, select the group to delete from the **Groups on this controller** list and click **Delete**.
2. You will now be asked if you want the users who belong to this group to continue belonging to it though it is not valid.

Click	To
Yes	Let the users of the group remain members to it even if it is no longer defined in the controller's UAS. This option might be useful if you plan to recreate the group, or copy the user's settings to another controller who has the group defined.
No	Delete the user's memberships to the group.
Cancel	Cancel the change and keep the group, with its user's memberships.

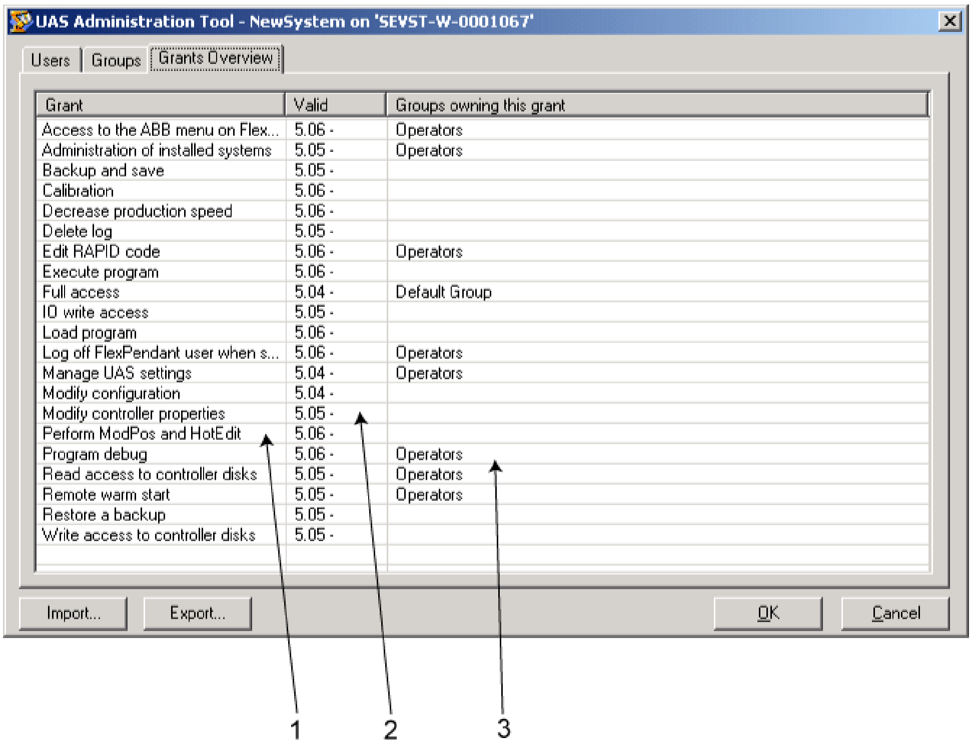
3. Click **OK**.

Giving grants to a group

1. On the **Groups** tab, select the group from the **Groups on this controller** list.
2. In the **Controller grants/Application grants** list, select the grants to give to the group.
3. Click **OK**.

12.7. UAS Grant Viewer

The Grants Overview dialog box



en0400001106

Parts

You can sort the information in all three columns by clicking the column heading.

1. The **Grant** column shows the name of the grant key.
2. The **Valid** column shows from which RobotWare versions the grant is valid.

   Systems with different RobotWare versions may use different grants. When you install a system on a controller, the grants for that system are added to the controller's grant list. Grants are never removed from the grant list but may be declared invalid if they are not applicable for a system. All this is handled automatically during installation of systems.
3. The **Groups owning this Grant** column shows which groups on this controller that have the grant.

Examples of common actions to perform

Action	Necessary grants
Rename the controller. (A restart of the controller is necessary.)	Modify controller properties Remote warm start
Change system parameters and load configuration files.	Modify configuration Remote warm start
Install a new system.	Administration of installed system
Perform a backup. (A restart of the controller is necessary.)	Backup and save Remote warm start

Continues on next page

Continued

Action	Necessary grants
Restore a backup. (A restart of the controller is necessary.)	Restore a backup Remote warm start
Load/delete modules.	Load program
Create new modules.	Load program
Edit code in RAPID modules.	Edit RAPID code
Save modules and programs to disk.	Backup and save
Start program execution from Task Window.	Execute program
Create a new I/O signal, that is, add a new instance of the type Signal. (A restart of the controller is necessary.)	Modify configuration Remote warm start
Set the value of an I/O signal.	I/O write access
Access to controller disks from File Management Window.	Read access to controller disks Write access to controller disks

Controller grants

Full access	This grant includes all controller grants, also new grants added in future RobotWare versions. The grant does not include any application grants.
Manage UAS settings	Gives access to read and write the UAS configuration, that is to read, add, remove and modify UAS users and groups.
Execute program	Gives access to perform the following: • Start/step program (stop is always allowed) • Move PP to Main • Execute service routines
Perform ModPos and HotEdit	Gives access to perform the following: • Modify or teach positions in RAPID code (ModPos) • During execution modify positions in RAPID code as single points or as a path (HotEdit) • Restore ModPos/HotEdit positions to original • Modify current value of any RAPID variable
Modify current value	Gives access to modify current value of any RAPID variable. This grant is a subset of the grant *Perform ModPos and HotEdit*. (Available from RobotWare 5.07.03.)
I/O write access	Gives access to perform the following: • Set I/O signal value • Set signal as simulated and remove simulation • Set I/O unit and bus as enabled/disabled
Backup and save	Gives access to perform a backup and to save modules, programs and configuration files.
Restore a backup	Gives access to restore backup and perform B-start.
Modify configuration	Gives access to modify the configuration database, that is to load configuration files, change system parameter values and add/delete instances.
Load program	Gives access to load/delete modules and programs.
Remote warm start	Gives access to perform warm start and shutdown (S-start) from a remote location. No grant is required to perform warm start via a local device, as for example the FlexPendant.

Continues on next page

Edit RAPID code	Gives access to perform the following: • Modify code in existing RAPID modules • Frame calibration (tool, workobj) • Commit ModPos/HotEdit positions to current values • Rename program
Program debug	Gives access to perform the following: • Move PP to routine • Move PP to cursor • HoldToRun • Activate/deactivate RAPID tasks • Request write access from the FlexPendant • Acknowledge Auto mode without restoring simulated I/O signals or deactivated tasks • Enable/disable non-motion execution
Decrease production speed	Gives acces to decrease speed from 100% in Auto mode. This grant is not required if speed is already below 100%, or controller is in Manual mode.
Calibration	Gives access to perform the following: • Fine calibrate mechanical unit • Calibrate base frame • Update/clear SMB data **Note!** Frame calibration (tool, wobj) requires the grant *Edit RAPID code*. Manual offset of mechanical unit calibration data and loading new calibration data from file require the grant *Modify configuration*.
Administration of installed systems	Gives access to perform the following: • Install new system • P-start • I-start • X-start • C-start • Select System • Install system from device This grant gives full FTP access, that is, the grant gives the same rights as *Read access to controller disks* and *Write access to controller disks*.
Read access to controller disks	Gives external read access to controller disks. This grant is only valid for explicit disk access, for example with an FTP client or the File Manager of RoboStudio. It is possible, for example, to load a program from /hd0a without this grant.
Write access to controller disks	Gives external write access to controller disks. This grant is only valid for explicit disk access, for example with an FTP client or the File Manager of RoboStudio. It is possible, for example, to save a program to the controller disk or perform a backup without this grant.
Modify controller properties	Gives access to set controller name, controller ID and system clock.
Delete log	Gives access to delete messages in the controller Event Log.
Revolution counter update	Gives access to update the revolution counter.

Continues on next page

Application grants

Access to the ABB menu on FlexPendant	Value **true** gives access to the ABB menu on the FlexPendant. This is the default value if a user does not have the grant. Value **false** means that the user cannot access the ABB menu when the controller is in Auto mode. The grant has no effect in Manual mode.
Log off FlexPendant user when switching to Auto mode	A user having this grant is automatically logged off from the FlexPendant when switching from Manual mode to Auto mode.

12.8. FlexPendant Viewer

Overview

FlexPendant Viewer is an add-in to RobotStudio that retrieves and displays a screenshot from the FlexPendant. The screenshot is generated automatically at the moment of the request.

Prerequisites

To install FlexPendant Viewer, the system must have the option *PC interface* installed.

The controller you want to retrieve screen shots from must be added to your robot view.

A FlexPendant must be connected to the controller. If no FlexPendant is currently connected (option *Hot plug* is installed and the jumper plug is used) then no screen shot can be retrieved.

Using FlexPendant Viewer

1. Make sure you are connected to the controller.
2. Click **FlexPendant Viewer**.

   A screen shot will be displayed in the workspace.
3. To reload the screen shot, click **Reload** in the workspace.
4. To set an automatic reload period for the screen shot, click on the menu **Tools**, point to **FlexPendant Viewer** and click **Configure**.

   Set the desired reload period and select the check-box **Activated**. Then click **OK**.

Results on the controller

The screenshot will automatically be saved as a file on the controller. When a new request is sent, a new screenshot is generated and saved, overwriting the previous file.

No message will be displayed on the FlexPendant.

12.9. Device Browser

Updating the tree view

1. To update the tree view, press **F5**.

Displaying the properties of a device

1. Select a device in the tree view to view its properties or values in the right-hand pane.

Displaying a trend

1. Select a device in the tree view and then double-click any property in the right-hand pane that has a numerical value to bring up a trend view.

   The trend view collects data at a rate of one sample per second.

Hiding, stopping, starting or clearing a trend

1. Right-click anywhere in the trend view and then click the appropriate command.

13 The Add-Ins Tab

13.1. Overview

The Add-Ins tab

The Add-Ins tab contains the control for PowerPacs and the VSTA.

13.2. Visual Studio Tools for Applications

Building an add-in

1. Click **Visual Studio Tools for Applications**.
2. Create a new project or open an existing one. Compose or edit as required.
3. To build the add-in, from the **Build** menu, click **Build**.

   The add-in will be loaded in RobotStudio and appear as a user add-in in the Add-In browser.
4. Save the project to your hard disk if you want the add-in to remain with the user. Otherwise, open a station, right-click in the Add-In browser and then click **Add to Station**.

14 The Context Menus

14.1. Add to Path

Creating a move instruction based on an existing target

1. Select the target for which to create the move instruction.
2. From the **Home** menu, in the **Path Programming** group, select the type of move instruction to create.
3. Click **Add to Path**.

   The move instruction will appear under the path node as a reference to the original target.

14.2. Align Frame Orientation

The Align Frame Orientation dialog box

Reference	Specify the frame or target for which you want to align the selected objects here.
Align Axis	The axis you specify here will be aligned as on the reference target/frame for all selected objects.
Lock Axis	The axis you specify here will not be changed on the selected objects by the align function, but will keep its orientation.

14.3. Align Target Orientation

Aligning target orientation

1. Select the targets whose orientation you wish to change.
2. Click **Align Target Orientation** to bring up a dialog box.
3. In the **Reference** box, specify the target whose orientation you want to use as reference, by first click in the box and then selecting the target either from the graphics view or the **Layout** browser.
4. In the **Align Axis** box, select the axis whose orientation you want to copy from the reference target to the selected ones.
5. In the **Lock Axis** box, select the axis to rotate the target around. The orientation of this axis will not be changed on the targets. For example, if the Z axis of all targets are orientated normally to the surface of the work piece and you want to keep it this way, you should lock the Z axis.
6. Click **Apply**.

TIP!

You can change the Align and Lock axis and click Apply again to reorientate the targets until you deselect them.

14.4. Attach to

Attaching an object

1. In the **Layout** browser, right-click the child object, click **Attach to** and click the parent object in the list.

Attaching an object by drag and drop

1. In the **Layout** browser, drag the child object to the parent object.
2. In the displayed message, click the corresponding button:

To	Click
attach the child object and move it to the attachment point	**Yes**
attach the child object and keep its position	**No**
not perform the attachment	**Cancel**

14.5. Auto Configuration

Setting a robot axis configuration for all targets in a path

1. In the **Paths&Targets** browser, select a path and then click **AutoConfiguration**. The robot now steps through each target in the path and sets the configurations.
2. If the first target in the path has no configuration assigned, the one closest to the robot's current configuration will be used. If the first target has a configuration assigned, the assigned one will be used. Thus, the result of the autoconfiguration may vary depending on the first target's configuration.

14.6. Check Reachability

Checking the reachability

1. In the **Paths&Targets** browser, select the task that contains the targets or move instructions to check.
2. Click **Reachability** to open a dialog box. Make sure that the **Check reachability for targets, paths, move instructions** check box is selected.

   Optionally, select or clear the **Check reachability for targets, paths, move instructions** check box for showing and hiding the reachability display.
3. In the **Layout** browser, select the objects whose reachability you want to check. If paths are selected, the reachability will be checked for all move instructions in each path.
4. In the **Reachability** dialog box, click **Add**.

   The frames for the selected objects will now change color based on their reachability status.

Color	Means
Green	The object can be reached.
Yellow	The object can be reached at its current position, but not at its current orientation
Red	The object cannot be reached at its current position.

14.7. Configurations

Manually setting a robot axis configuration for single targets

1. In the **Paths&Targets** browser, select a target and then click **Configurations** to bring up a dialog box.
2. If more than one configuration solution exist, examine them by clicking them, one at a time.

   The position of the robot with the selected configuration will be displayed in the graphics window, and the joint values for the configuration will be displayed in the joint values list below the configurations list.

   In most cases, selecting a configuration similar to the previous one is the best choice.
3. Select the configuration to use and click **Apply**.

14.8. Control Panel

The Control Panel dialog box

Operation Mode	This group contains the three operational modes of the controller represented by option buttons.
Auto	This option corresponds to the Auto mode on the FlexPendant. Moving between the **Auto** and **Manual Full Speed** options must proceed via the **Manual** option.
Manual	This option corresponds to the Manual mode on the FlexPendant.
Manual Full Speed	This option corresponds to the Manual 100% mode on the FlexPendant. Moving between the **Auto** and **Manual Full Speed** options must proceed via the **Manual** option.
Push Motors	Click this button to turn on the motors.
Enable Device	In a manual mode, click this button to simulate holding the enabling device to turn on the motors.
Release Device	In a manual mode, click this button to turn off the motors.
Reset Emergency Stop	If the controller enters the emergency stop state, click this button to reset the state.

14.9. Convert Frame to Workobject

Converting a frame to a workobject

1. In the **Layout** browser, select a frame.
2. Click **Convert Frame to Workobject**. The new workobject will appear in the **Paths&Targets** browser.
3. Optionally, rename or edit the workobject in any way.

14.10. Convert to Move Circular

Prerequisites

At least two targets, the via-point target and the end point target, must have been created.

A path containing at least the via-point target and the end point target, in correct order, must have been created.

Converting to Move Circular

1. In the **Paths&Targets** browser, expand the path node that contains the move instruction to be converted.
2. Select the move instruction that contains the via-point of the circular motion together with the succeeding move instruction, which will serve as the end point. You can select several instructions by holding down the **SHIFT** key while clicking the instructions.
3. Click **Convert to Move Circular**. The two selected move instructions will be converted to a circular move instruction, which includes the via-point and the end point.

TIP!

To convert two move instructions to a circular motion, you can also select and right-click both move instructions at once and then click **Convert to Circular**.

14.11. Copy / Apply Orientation

Copying and applying an orientation

1. In the browser, select the object or target from which to copy the orientation.
2. On the **Modify** menu, click **Copy Orientation**.
3. In the browser, select the object or target to which to apply the orientation.
4. On the **Modify** menu, click **Apply Orientation**. This can be performed on several targets or a group of selected targets.

14.12. Detach

Detaching an object

1. In the **Layout** browser, right-click the attached object (child) and then click **Detach**. The child will be detached from the parent and return to its position before the attachment.

14.13. Execute Move Instruction

Prerequisites

The move instruction must exist.

A virtual controller must be running for the robot with the move instruction.

Executing a move instruction

1. In the **Paths&Targets** browser, browse to the motion instruction to execute through the **Controller**, **Tasks** and **Paths** nodes.
2. Click **Execute move instruction**. The TCP of the active robot will move from the current location to the motion instruction according to the programed motion properties. If the target for the motion instruction does not have a stored configuration, the robot will use the configuration nearest the current one.

14.14. Graphic Appearance

Overview

With the graphic appearance dialog box you set the graphic properties for an individual object. The settings made here override the generic settings made in the options dialog box. The dialog box contains one view group where you select the part of the object to affect, and three tabs with settings.

Graphic Appearance: The View group

Selection	Select the parts of the object whose appearance you wish to change. When **Body** or **Surface/Curve** is selected, you select the object to work with in the preview display. **NOTE!** To change the appearance of a part: 1. On the **Modify** menu, click **Graphic Appearance**. A color box with the following options appears - Metals, Light colors, medium colors, Dark colors. 2. Select any of the options to change the appearnace.

Graphic Appearance: The Rendering tab

Backface culling	Select this check box to view only the front sides (the direction of the positive normal) of the faces in the model. This improves the graphics performance and indicates the directions of the faces in the model. Clear this check box to also display the back sides of the faces in the model. This ensures that the model is displayed correctly even if the faces are in the wrong direction. If a model with faces in the wrong direction is used for programming, you might get unpredicted results.
Twosided lightning	Select this check box to light the object from two sides. Clear this check box to light the object from one side.
Flat shade	Select this check box to change the appearance of shades.
Flip normals	Click this button to change the direction of all faces on the model.
Line width	Specify the width of lines in the object here.
The detail level options	Select the detail level of the model here. Only the levels selected in the **Options** dialog box when the object was created or imported are available.

Graphic Appearance: The Colors tab

Simple Color	Click this color box to select another color for the object.
Transparency	Use this slider to control the transparency of the object.
The color boxes	Set the color of the object for different light situations here.
Shininess	Specify the reflectiveness of the object here.

Continues on next page

Graphic Appearance: The Textures tab

Effect	Specifies the type of graphic effect to be used on the selected part. • Basic : Simple texturing/environment mapping or just a solid color. • Anisotropic: Brushed metal effect simulating an anisotropic light model • Bump Mapping: Texture that specifies the bumpiness of the surface • Parallex Mapping: Improved version of bump mapping **NOTE!** Anisotropic, Bump Mapping and Parallex Mapping requires DirectX9-class hardware to be correctly displayed in the 3D window.
Base texture	Specifies the basic structure of the selected part. It is a standard 24-bit image displayed on a 3D surface.
Environment map	Provides a highly reflective appearance to the surface.
Normal/Bump Map	Specifies a texture that defines the bumpiness of the surface. **NOTE!** This option is enabled only during Bump mapping and Parallex mapping effect.
Blend with material	Select this check box to blend the texture with the colors and properties specified on the Material tab.
Swap u/v	Click this button to swap the horizontal and vertical directions of the texture.
Modify	Select along which directions the commands listed below shall be applied. **u** is the horizontal axis of the texture. **v** is the vertical axis of the texture.
Normalize	Click this button to set the ratio between the dimensions of the object and the texture to 1.
Flip	Click this button to invert the coordinates along the selected axes. This is the same as mirroring around the other axis.
Stretch	Click this button to stretch the texture along the selected axes.
Shrink	Click this button to shrink the texture along the selected axes.
Shift <	Click this button to move the texture along the selected axes.
Shift>	Click this button to move the texture along the selected axes.

14.15. Interpolate Path

Reorienting targets in a path by interpolation

1. In the **Layout** browser or the graphics window, select the path with the targets to reorient.
2. Click **Interpolate Path**. to bring up a dialog box.
3. With the **Interpolate type** options, select whether to use **Linear** or **Absolute** interpolation.

   Linear interpolation distributes the difference in orientation evenly, based on the targets positions along the length of the path. By contrast, absolute interpolation distributes the difference in orientation evenly, based on the targets' sequence in the path.
4. If using the **Select Start/End** option, select the start and end targets for the interpolation in the **Start target** and **End target** boxes, respectively.
5. Optionally, with the **Lock Axis** options, select an axis to lock.
6. Click **Apply**.

14.16. Invert

Inverting the direction of a face

1. Right-click in the **Modeling** browser, point to **Filter** and make sure that both **Show Bodies** and **Show Faces** are selected.
2. In the **Modeling** browser, expand the node for the object and browse down to and select the face which direction you want to invert.
3. In the **Modeling** browser, expand the node for the object and browse down to and select the face whose direction you want to invert.
4. Click **Invert**. If the option backface culling is activated, the face will now shift from visible to not visible, or the other way around, depending on from which direction you view the face. If backface culling is deactivated, there will be no visible indication that the direction of the face has been inverted.

14.17. Jump to Target

Jumping to a target

1. In the **Paths&Targets** browser, browse to the target to jump to through the **Controller**, **Tasks** and **WorkObjects** nodes.
2. Click **Jump to target**.

   If the target has a valid configuration for the robot axes stored, the active TCP of the robot will immediately be positioned at the target. If no valid configuration is stored, the **Select Robot configuration** dialog box is displayed.
3. In the **Select Robot Configuration** dialog box, select a suitable configuration solution and click **Apply**. The selected configuration is now stored with the target.

NOTE!

You can deactivate the configuration check when jumping to targets. The robot will then use the configuration solution closest to the current one when reaching the target. For more information, see *RobotStudio Options on page 178*.

14.18. Linked Geometry

Overview

The Linked Geometry feature allows you to load geometry from a shared repository. If the source file is updated, then the station will be updated with a single click.

Adding Link

You can add a link to a geometry in two ways:

1. In the **Home** tab, click **Import Geometry** to open a dialog box.

   Select the option **Link to Geometry**.
2. In the **Layout** browser, right-click an existing part in the station and select **Add Link**.

   A dialog box opens where you can select the CAD file to be linked.

Editing Link

To edit an existing link:

1. In the **Layout** browser, right-click an existing part in the station.
2. Select the option **Link to Geometry** and click **Edit Link**.

Deleting Link

To delete an existing link:

1. In the **Layout** browser, right-click an existing part in the station.
2. Select the option **Link to Geometry** and click **Delete Link**.

Updating Linked Geometry

To update a linked geometry:

1. In the **Layout** browser, right-click an existing part in the station, component group or the station.
2. Select the option **Link to Geometry** and click **Update Linked Geometry**.

   The update result is displayed in the output window.

   NOTE! When you select a component group or a station, all linked geometries within the group or station is updated.

   NOTE! If the timestamp on the file is newer than the timestamp stored in the station, all corresponding parts will be updated from the source location.

14.19. The Library Group

Modifying a library component

1. In the **Layout** browser, select the library you wish to modify.
2. Click **Disconnect Library**.
3. Select the library and then make any modifications to it.
4. Select the modified library, and then click **Save As Library.**

14.20. Mechanism Joint Jog

Jogging the joints of a robot

1. In the **Layout** browser, select the robot.
2. Click **Mechanism Joint Jog** to bring up a dialog box.
3. Each row in the **Jog Joints** dialog box represents a joint of the robot. Jog the joints either by clicking and dragging the bar at each row, or by using the arrows to the right of each row.

   Set the length of each step in the **Step** box.

The Mechanism Joint Jog dialog box

Joint	Move the joints of the objects by dragging the slider on the row corresponding to each joint. Alternatively, click the buttons to the right of the row, or type a value.
Cfg	The current configuration value.
TCP	The current position of the TCP.
Step	Specify the length of the joint movements for each click of the buttons to the right of each joint row.
External Axis	If the robot uses external axes, you can select an axis to jog from this list. The external axes must belong to the same task as the object you are jogging for occurring in this list. If no external axes are present in the same task, this list is not available.
Lock TCP	Select this check box to reposition the robot according to the jogging of the external axis. For track-external axes the robot will reposition so that the TCP is locked relative to the world coordinate system. For positioner-external axes the robot will reposition so that the position of the robot’s TCP is locked relative to the attachment point of the positioner. The robot will move with the positioner the same way as when using multi-robot jog. If no external axes are present in the same task, this check box is not available.
External axes joint	Move the joint of the external axes by dragging the slider on the row corresponding to each joint. Alternatively, click the buttons to the right of the row, or type a value. If no external axes are present in the same task, this check box is not available.

Jogging a conveyor

1. Create **Empty Path**. See *Empty Path on page 201*.
2. In the **Layout** browser, select the conveyor.
3. Right-click **Conveyor Mechanism** and select **Mechanical Joint Jog**.

   The Joint Jog dialog box appears.
4. Jog the conveyor by moving the slider and click **Teach Instruction**.

   A move instruction is added to the path.

Continues on next page

NOTE!

When you jog the conveyor mechanism, objects on the conveyor are also moved.

- If you jog the conveyor mechanism beyond the maximum distance, the workobject will be dropped.
- If you jog the conveyor mechanism beyond the zero position, the workobject that belongs to the first part is attached to the conveyor attachment point.

If the workobject is dropped in Teach Mode, you can jog the conveyor backwards to connect it again.

14.21. Mechanism Linear Jog

Jogging the TCP of a robot by using the Linear Jog dialog box

1. In the **Layout** browser, select the robot.
2. Click **Mechanism Linear Jog** to bring up a dialog box.
3. Each row in the **Linear Jog** dialog box represents a direction or rotation for the TCP. Jog the TCP along the preferred direction or rotation, either by clicking and dragging the bar at each row, or by using the arrows to the right of each row.
4. From the **Reference** list, you can select the coordinate system that you want to jog the robot relative to.
5. In the **Step** box, specify the step movement per deg/rad.

14.22. Mirror Path

The Mirror Path dialog box

Duplicate	Select this option to keep the existing path when mirroring.
Replace	Select this option to remove the existing path after mirroring.
X-Y, X-Z and **Y-Z**	Select the plane to mirror the path around with these options. The plane is defined by the selected axes and position of the reference coordinate system selected below.
Reference	Select the frame or coordinate system to define the mirror plane in. To use another frame than any of the predefined ones, select **Select Frame** from the list and specify the frame in the box below.
Select Frame	If **Select Frame** is used as Reference frame, specify the frame to use here by first clicking in the box and then selecting the frame from the graphics window or the **Layout** browser.
Flip axis X/YZ	Select one of these options to mirror the orientation of the targets. When any of these are selected, the robot will approach the targets in a mirrored way. The axis you select will change the most for achieving the mirrored orientation, while the other one will be kept as near to its current direction as possible. The axis that is set to the robot's approach vector cannot be selected.
Keep orientation	Select this option to keep the orientation of the targets. When selected, the robot will go to the mirrored position, but approach the target from the same direction as for the original target.
Mirror Robot Configuration	Select this option to also mirror the robot axis configuration for the targets. Selecting this option will mirror the robot's motions completely. For using this option, the following conditions must be met: • The Reference frame must be set to *Baseframe*. • The Mirror plane must be set to X-Z. • The tool of each move instruction must have its TCP in the X-Z plane of *tool0*. • All targets in the path must have robot axis configuration set. • The virtual controller must be running.
More / **Less**	Click this button to show or hide the commands for naming and location of generated targets and paths.
New path name	Specify the name of the path that will be generated by the mirroring here.
Target prefix	Specify a prefix for the targets that will be generated by the mirroring here.
Receiving robot	Specify the robot task in which the new targets and path shall be created.
Receiving work object	Specify the work object in which the new targets shall be created.

14.23. Modify Curve

Extending a curve with a straight line in the direction of the curve's tangent

1. Click **Modify curve** to bring up a dialog box.
2. Select **Extend** as modify method and then click the curve segment, close to the appropriate vertex.

   Note that when you rest the pointer over the curve the endpoint closest to the pointer is highlighted. This is the endpoint that will be selected for extension when you click the curve.
3. In the **Distance from endpoint** box, enter the length of the extension. In the graphics window a yellow line displays a preview of the extension.
4. Click **Apply**.

Joining curves

1. Click **Modify curve** to bring up a dialog box.
2. Select **Join** as modify method and then click the curves to join in the graphics window. The curves may be either intersecting or adjacent to be joined.

   The **Selected curves** list displays the curves that will be joined. To remove a curve from the list, select the list entry and press the DEL key.
3. In the **Tolerance** list, enter a value in millimeters. Adjancent curves whose end points lie within the tolerance will be valid for the operation.
4. Click **Apply**.

Projecting curves on a surface

1. Click **Modify curve** to bring up a dialog box.
2. Select **Project** as modify method and then click the curves to project in the graphics window.

   Note that when you rest the pointer over the curve, the projection direction is displayed. The project direction is always the negative Z direction of the User Coordinate System. To change the projection direction, create a new frame with the desired orientation and set it as user coordinate system.

   The **Selected curves** list displays the curves that will be projected. To remove a curve from the list, select the list entry and press the DEL key.
3. Click in the **Target bodies** list and then click the bodies to project on in the graphics window. The bodies must be in the projection direction and be big enough to cover the projected curves.

   To remove a bodies from the list, select the list entry and press the DEL key.
4. Click **Apply**. A new curve will now be created in a new part, wrapped around the surface of the selected bodies.

Continues on next page

Reversing curves

1. Click **Modify curve** to bring up a dialog box.
2. Select **Reverse** as modify method and then click the curves to reverse in the graphics window.

   Note that when resting the pointer over a curve, the current direction of the curve is displayed by yellow arrows.

   The **Selected curves** list displays the curves that will be reversed. To remove a curve from the list, select the list entry and press the DEL key.
3. Click **Apply**. The curves will now be reversed.

Splitting a curve

1. Click **Modify curve** to bring up a dialog box.
2. Select **Split** as modify method and then click the curve at the point to split at. Only open curves can be split.

   Note that when you rest the pointer over the curve, point of the split is highlighted. This point is affected by the current snap mode setting.
3. Click **Apply**. The curve will now be split to two separate curves in the same part.

Trimming a curve

1. Click **Modify curve** to bring up a dialog box.
2. Select **Trim** as modify method and then click the curve segment to trim.

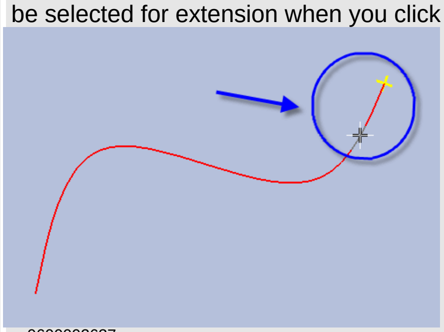
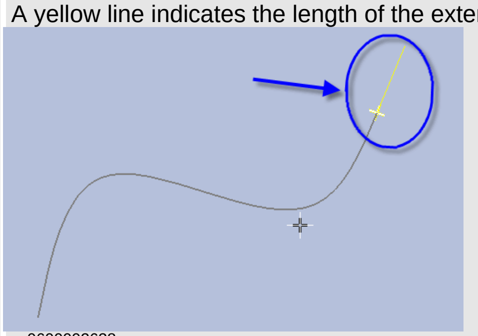
   Note that when you rest the pointer over the curve, the closest vertices are highlighted. The segment between these points will be trimmed.
3. Click **Apply**. The selected part of the curve will now be removed.

Generic content of the Modify Curve dialog box

Extend	Extend a curve at any vertex with a straight line in the direction of the curve’s tangent.
Join	Join two or more curves into one. The original curves will be deleted when joining curves.
Project	Project a curve onto a surface or a body, creating a new curve on the target part.
Reverse	Reverse the direction of curves.
Split	Split a curve in two bodies. Only open curves can be split.
Trim	Cut a segment of a curve between intersection or end points.

Continues on next page

Extend-specific information

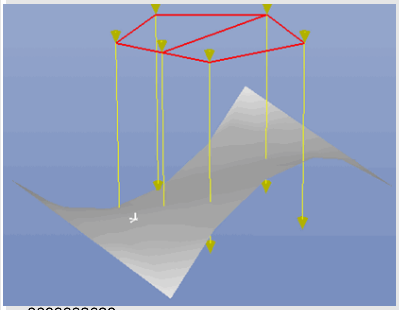
Selected curve	Display the name of the curve to extend. Select the curve by clicking it in the graphics window. Note that when you rest the pointer over the curve the endpoint closest to the pointer is highlighted. This is the endpoint that will be selected for extension when you click the curve.  xx0600002637
Endpoint to extend	Display the position of the endpoint to extend. To change the endpoint, select the curve again, but click it closer to the other endpoint.
Distance from endpoint	Type the length of the extension here. You can also click a point in the graphics window to set the length. A yellow line indicates the length of the extension.  xx0600002638

Join-specific information

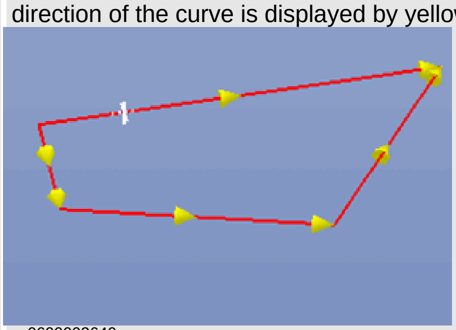
Selected curves	Display the names of the curves to join. Select the curves by clicking them in the graphics window. To remove a curve from the list, select the list entry and press the DEL key. The curves may be either intersecting or adjacent to be joined.
Tolerance	Specifies the distance within which the end points of adjacent curves may lie to be joined.

Continues on next page

Project-specific information

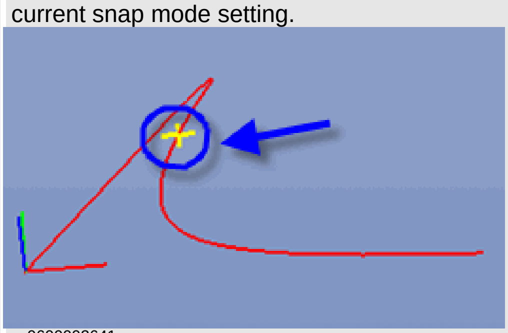
Selected curve	Display the name of the curves to project. Select the curves by clicking them in the graphics window. Note that when you rest the pointer over the curve the projection direction is displayed. The project direction is always the negative Z direction of the User Coordinate System. To change the projection direction, create a new frame with the desired orientation and set it as user coordinate system.  xx0600002639
Target bodies	Display the name of the bodies to project the curve on. Select the bodies by first clicking in the box and then clicking the bodies in the graphics window.

Reverse-specific information

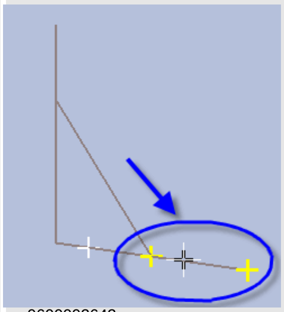
Selected curves	Display the names of the curves to reverse. Select the curves by clicking them in the graphics window. Note that when resting the pointer over a curve the current direction of the curve is displayed by yellow arrows.  xx0600002640

Continues on next page

Split-specific information

Selected curve	Display the name of the curve to split. Select the curve by clicking it in the graphics window. Note that when you rest the pointer over the curve, the point where the split will be is highlighted. This point is affected by the current snap mode setting.  xx0600002641
Point on curve	These boxes show the position of the split point. To change the split point, select the curve again, but click another part of it.

Trim-specific information

Selected curve	Display the name of the curve to trim. Select the curve by clicking it in the graphics window. Note that when you rest the pointer over the curve the closest vertexes are highlighted. It is the segment between those points that will be cut.  xx0600002642 Trim only works on single curves with intersection points. If you want to trim a curve that intersects with another curve, first join the two curves.
First section endpoint	Display the position of the first trimming point.
Second section endpoint	Display the position of the second trimming point.

14.24. Modify External Axis

Modifying external axis positions in targets

1. Select the targets (one or several) you want to modify, either in the **Layout** browser or in the graphics window. If you select several targets, the values you specify will be applied to all selected targets.
2. Click **Modify External Axis** to bring up a dialog box.
3. Edit the values of the axis by performing any of the following:

Action	Description
Type a new position value for an axis	In the **Eax** column, select the value of the external axis you want to edit, and enter the new value.
Jog the axis to the new position	Use the arrow buttons to the left of the Joint Values column for jogging the axis. Then click the right arrow button between the Joint values column and the Eax column to transfer the current joint value to the Eax value.

4. Click **Apply**.

The Modify External Axis dialog box

<	Jog the joint of the external axis corresponding to each row by clicking the **<** button.
>	Jog the joint of the external axis corresponding to each row by clicking the **>** button.
value box	Enter the axis value for the corresponding joint of the external axis in the value box.
<-	With the arrow left button, you transfer the value from the **Eax** box to the corresponding value box.
->	With the arrow right button, you transfer the value from the value box to the corresponding **Eax** box.
Eax	Specify the value of the corresponding joint of the external axis.

14.25. Modify Instruction

Modifying an instruction

1. In the **Paths&Targets** browser, select the instruction you want to modify. If you want to apply the same properties to several instructions, press the **CTRL** key and select them.
2. Click **Modify Instruction** to bring up a dialog box.
3. For move instructions, select joint or linear motion in the **Motion type** list.
4. In the **Instruction Argument** group, modify the values for the instruction.

   For details about each argument, see the selected instruction in the *RAPID Reference Manual*. For an overview of the arguments for move instructions, see below.
5. When you have finished modifying, click **Apply**.

Arguments for move instructions

The table below is an overview of common arguments for move instructions. For detailed information about the arguments, see the selected instruction in the *RAPID Reference Manual*.

To set the	Use
subsequent instructions to be executed at once.	**\Conc**
destination target for the instruction target.	**ToPoint**
speed for the tool center point, the tool reorientation and external axes.	**Speed**
velocity of the TCP in mm/s directly in the instruction (it will be substituted for the corresponding speed data).	**\V**
total time in seconds during which the robot moves (it will be substituted for the corresponding speed data).	**\T**
size of the generated corner path.	**Zone**
position accuracy of the robot TCP directly in the instruction (the length of the path will be substituted for the corresponding zone specified in the zone data).	**\Z**
tool used for the movement (the TCP of this tool will be positioned at the destination target).	**\Tool**
workobject to which the robot position in the instruction belongs.	**\Wobj**

14.26. Modify Mechanism

The Modify Mechanism dialog box

See *The Modify Mechanism dialog box on page 251*.

14.27. Modify Tooldata

Modifying tooldata

1. In the **Layout** browser, select the tooldata you want to modify.
2. Click **Modify Tooldata** to bring up a dialog box.
3. In the **Misc Data** group:
   - Modify the **Name** of the tool.
   - Select if the tool is to be held by the robot in the **Robot holds tool** list.
4. In the **Tool Frame** group:
   - Modify the **Position x, y, z** of the tool.
   - Modify the **Rotation rx, ry, rz** of the tool.
5. In the **Load Data** group:
   - Enter a new **Weight** for the tool.
   - Modify the **Center of gravity** for the tool.
   - Modify the **Inertia** for the tool.
6. In the **Sync Properties** group:
   - In the **Storage type** list, select **PERS** or **TASK PERS**. Select **TASK PERS** if you intend to use the tooldata in MultiMove mode.
   - In the **Module** list, modify the module in which to declare the tooldata.
7. Click **Apply**.

14.28. Modify Workobject

Modifying a workobject

1. In the **Layout** browser, select the workobject you want to modify.
2. Click **Modify Workobject**. to bring up a dialog box.
3. In the **Misc Data** group, modify the values for the workobject:
    - Enter a **Name** for the workobject.
    - In the **Robot holds workobject** list, select **True** or **False**. If you select **True**, the robot will move the work piece instead of the tool.
    - In the **Moved by mechanical unit** list, select the mechanical unit with which the robot movements are coordinated. This setting is only valid when **Programmed** has been set to **False**.
    - In the **Programmed** list, select **True** or **False**. **True** means that the workobject will use a fixed coordinate system, and **False** that a movable coordinate system (that is, coordinated external axes) will be used.
4. In the **User Frame** group, do one of the following:
    - Modify the user frame by entering values for the **Position x, y, z** and the **Rotation rx, ry, rz** for the workobject. Click in one of these boxes, and then click the position in the graphics window to transfer the values.
    - Modify the user frame by using the **Frame by points** dialog box, see *Frame from Three Points on page 193* .
5. In the **Object Frame** group, do one of the following:
    - Modify the object frame by selecting values for **Position x, y, z** and **Rotation rx, ry, rz** for the workobject.
    - Modify the object frame by using the **Frame by points** dialog box.
6. In the **Sync Properties** group, modify the values for the workobject:
    - In the **Storage type** list, select **PERS** or **TASK PERS**. Select **TASK PERS** if you intend to use the workobject in MultiMove mode.
    - In the **Module** list, select the module in which to declare the workobject.
7. Click **Apply**.

NOTE!

If you change the position of a workobject that is used in a program, you have to synchronize the affected paths to the virtual controller; otherwise, the program will not be updated.

14.29. Move Along Path

Prerequisites

At least one path must have been created in the station.

A virtual controller must be running for the robot to move along the path.

Moving along a path

1. In the **Paths&Targets** browser, select the path to move along.
2. Click **Move along path**. In the graphics window, the robot will move along the path.

14.30. Move to Pose

Prerequisites

At least one joint position must be defined.

Only one mechanism may be selected at a time.

Moving to a pose

1. In the **Layout** browser, select one mechanism to move.
2. Click **Move to Pose** and then click one of the available poses. In the graphics window, the mechanism will move to the pose.

14.31. Place

Placing an item

1. Select the item you want move.
2. Click **Place** and then click one of the commands to bring up a dialog box.

If you want to move the item	Choose
from one position to another without affecting the orientation of the object. Select the axes to be affected.	**One Point**
according to the relationship between a start and a finish line. The object will move to match the first point, then it will rotate to match the second point.	**Two points**
according to the relationship between a start plane and a finish plane. The object will move to match the first point, then it will rotate to match the third point.	**Three Points**
from one position to a target or frame position and simultaneously change the orientation of the object according to the frame orientation. The position of the object changes according the orientation of the to-point coordinate system.	**Align Object by Frame**

3. Set the reference coordinate system you want to use.
4. Click the points in the graphics window to transfer values to the from-point boxes to the to-point boxes. For detailed information, see the tables below.
5. Click **Apply**.

The Place Object by One Point dialog box

Reference	Select the reference coordinate system to which all positions or points will be related.
Primary Point - From	Click in one of these boxes, and then click the primary point in the graphics window to transfer the values to the **Primary Point - From** boxes.
Primary Point - To	Click in one of these boxes, and then click the primary point in the graphics window to transfer the values to the **Primary Point - To** boxes.
Translate along these axes	Select whether the translation is to be performed along the **X**, **Y** or **Z** axis, or several of the axes.

The Place Object by Two Points dialog box

Reference	Select the reference coordinate system to which all positions or points will be related.
Primary Point - From	Click in one of these boxes, and then click the primary point in the graphics window to transfer the values to the **Primary Point - From** boxes.

Continues on next page

Primary Point - To	Click in one of these boxes, and then click the primary point in the graphics window to transfer the values to the **Primary Point - To** boxes.
Point on X-Axis - From	Click in one of these boxes, and then click the point on the x axis in the graphics window to transfer the values to the **Point on X-Axis - From** boxes.
Point on X-Axis - To	Click in one of these boxes, and then click the point on the x axis in the graphics window to transfer the values to the **Point on X-Axis - To** boxes.
Translate along these axes	Select whether the translation is to be performed along the **X**, **Y** or **Z** axis, or several of the axes.

The Place an Object by Three Points dialog box

Reference	Select the reference coordinate system to which all positions or points will be related.
Primary Point - From	Click in one of these boxes, and then click the primary point in the graphics window to transfer the values to the **Primary Point - From** boxes.
Primary Point - To	Click in one of these boxes, and then click the primary point in the graphics window to transfer the values to the **Primary Point - To** boxes.
Point on X-Axis - From	Click in one of these boxes, and then click the point on the x axis in the graphics window to transfer the values to the **Point on X-Axis - From** boxes.
Point on X-Axis - To	Click in one of these boxes, and then click the point on the x axis in the graphics window to transfer the values to the **Point on X-Axis - To** boxes.
Point on Y-Axis - From	Click in one of these boxes, and then click the point on the y axis in the graphics window to transfer the values to the **Point on Y-Axis - From** boxes.
Point on Y-Axis - To	Click in one of these boxes, and then click the point on the y axis in the graphics window to transfer the values to the **Point on Y-Axis - To** boxes.
Translate along these axes	Select whether the translation is to be performed along the **X**, **Y** or **Z** axis, or several of the axes.

The Place Object with Frame dialog box

Select Frame	Specify the name of the frame with which you want to place the object.

14.32. Remove Unused Targets

Removing unused targets

1. In the **Paths&Targets** browser, select either the *Controller* node or the *Task* node from which you wish to remove the unused targets, and then click **Remove Unused Targets**.
2. To the question “**Are you sure you want to remove unused targets?**”, answer **Yes**. All targets that not are used by any move instructions are now removed.

14.33. Rename Targets

Renaming targets

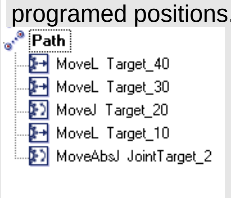
1. In the **Paths & Targets** browser, select the targets to rename.

   To rename all targets in one or several paths, select the paths that contain the targets.
2. Click **Rename targets** to bring up a dialog box.
3. In the **Target Prefix** box, enter a text string to precede the target numbers.
4. Optionally, in the **Increment** box and the **Start with** box, change the numbering series for the target names.
5. Optionally, in the **Target Suffix** box, enter a text string to follow the target numbers.
6. Click **Apply**.

14.34. Reverse Path

The commands

<table>
<tr>
<td>Simple</td>
<td>Here you reverse only the target sequence. The new path will keep the move instruction for each path segment and just reverse the programed positions.



xx0500002041

Note that move instructions are not changed, just the targets. Even the MoveAbsJ instruction to the joint target is preserved, but placed last.</td>
</tr>
<tr>
<td>Advanced</td>
<td>Both target sequence and move instructions are reversed in a way that corresponds to recording the robot movements and playing the movie backwards. For example, if the robot used a linear motion to move from a target, it will use a linear motion to move to the target after the reversal.

xx0500002042

Note that the move instructions have changed together with the targets. For example, in the original path, a joint motion was used to reach target 20 and a linear motion to leave it. After the reversal there is a linear motion to the target and joint motion from it.

Also, note that the jointtarget has been converted into an ordinary target; otherwise, it would not be possible to program a linear motion to that position.</td>
</tr>
</table>

14.35. Rotate

Rotating an item

1. Select the item you want to rotate.
2. Click **Rotate** to bring up a dialog box.
3. Select the reference coordinate system you want to use:

If you want to move the item	Select
absolute in the coordinate system of the station	**World**
relative to the coordinate system of its parent	**Parent**
relative to its own coordinate system	**Local**
relative to the user-defined system	**UCS**
relative to an axis defined by two points	**User defined axis**

4. Specify the rotation of the item in the **Rotate around x, y, z** by first clicking in one of the boxes, and then click the center position in the graphics window to transfer the values.
5. If you have selected the coordinate system **User defined axis**, specify the **Axis start point x, y, z** and the **Axis end point x, y, z**.
6. Specify the **Rotation** of the item and the axis around which the rotation is to occur.
7. Click **Apply**.

14.36. Rotate Path

Rotating a paths

1. In the **Layout** browser or the graphics window, select the paths to rotate.
2. Click **Rotate path** to bring up a dialog box.
3. In the **Reference frame** list, select the frame to rotate the paths around.

Select	To
World	rotate around the station's world coordinate system
Baseframe	rotate around the robot's baseframe
UCS	rotate around a frame or target that previously has been set to User Coordinate System.
Select Frame	rotate around an existing target or frame other than the listed ones. When using **Select Frame**, specify the frame to rotate around further down.

4. If **Selected frame** was selected in the **Reference frame** list, specify a frame or target in the text box by clicking in the box and then selecting the frame in the graphics window.
5. With the **Rotation axis** options, select the axis of the frame to rotate around.
6. In the **Rotation angle** box, enter the rotation.
7. Click **Apply**.

14.37. Save Module As

Saving a module

1. Click **Save Module As** to bring up a dialog box.
2. Browse to the location of the module to be loaded to your station and click **Save**.

14.38. Save Program As

Saving a program

1. Click **Save Program As** to bring up a dialog box.
2. Browse to the location where you want to save your program, and click **Save**.

14.39. Set Local Origin

Setting the origin of the local coordinate system

1. If the object you want modify is a library component, first disconnect it from the library.
2. In the **Layout** browser or the graphics window, select the part to modify.
3. Click **Set Local Origin** to bring up a dialog box.
4. In the **Set Local Origin** dialog box, select the reference coordinate system you want to use:

If you want to move	Select
relative to the part's current local coordinate system	**Local**
relative to the coordinate system of its parent	**Parent**
absolute in the coordinate system of the station	**World**
relative to a user-defined coordinate system	**UCS**

5. In the **Position X, Y, Z** boxes, either type the new position or, select it by first clicking in one of the value boxes and then clicking the point in the graphics window.
6. Type the **Orientation**.
7. Click **Apply**.

14.40. Set Normal to Surface

Setting the target orientation normal to a surface

1. In the **Paths & Targets** browser, select the target to modify.
2. Click **Set Normal To Surface** to bring up a dialog box.
3. On the **Selection Level** toolbar, set the selection level.
   - To align the target to a specific surface, set the selection level to **surface**.
   - To align the target to a specific point at the surface, set the selection level to **part**.
4. In the graphics window, click the reference surface. This will transfer the name of the part or surface to the **Surface** box.
5. In the **Approach Direction**, click the button for the axis to be used as the approach direction.
6. To set the distance between the surface and the target in the approach direction, specify an **Offset** value.
7. Click **Apply**.

14.41. Set Position

Positioning an item

1. Select the item you want to move.
2. Click **Set Position** to bring up a dialog box.
3. In the **Set Position** dialog box, select the reference coordinate system you want to use:

If you want to move the item	Select
relative to its own coordinate system	**Local**
relative to the coordinate system of its parent	**Parent**
absolute in the coordinate system of the station	**World**
relative to a user-defined coordinate system	**UCS**

4. In the **Position X, Y, Z** boxes, either type the new position, or select it by first clicking in one of the value boxes and then clicking the point in the graphics window.
5. Specify the **Orientation** for the item.
6. Click **Apply**.

14.42. Tool Compensation

Offsetting a path to compensate for tool radius

1. In the **Paths&Targets** browser or the graphics window, select the path.
2. Click **Tool Compensation** to bring up a dialog box.
3. In the **Distance** box, enter the size of the compensation (normally, the tool radius).
4. Using the **Direction** options, select whether the new path shall be on the left or the right side of the current path.
5. Click **Apply**.

14.43. Translate Path

Translating a path

1. In the **Paths&Targets** browser or the graphics window, select the paths to translate.
2. Click **Translate path** to bring up a dialog box.
3. In the **Reference frame** list, select the coordinate system to use as reference for moving the paths.

Select	To
World	move relative to the origin of the world coordinate system
Base Frame	move relative to the origin of the robot's baseframe
UCS	move relative to the origin of a frame or target that previously has been set to User Coordinate System.
Select Frame	move relative to the origin of an existing target or frame other then the listed ones. When using **Select Frame**, specify the frame to use further down.
Point to Point	move the path from one point to another without specifying any coordinate system.

4. If **Select frame** was selected in the **Reference frame** list, specify a frame or target in the text box by clicking in the box and then selecting the frame from the graphics window.
5. In the **Translation vector** box, specify the distance to move the path along the X, Y and Z axes of the reference frame.

   Translation vector is applicable only if a reference frame is used. If **Point to Point** is used as reference, specify the start and end points for the translation, instead. To do this, click in one of the boxes for the point to specify and then select the point in the graphics window, or type the coordinates of the point.
6. Click **Apply**.

14.44. View Robot at Target

Viewing a robot at a target

1. Click **View Robot at Target**.
2. Select a target, either in the **Paths&Targets** browser or in the graphics window.
3. The robot will be shown at each selected target whenever a target is selected. By stepping through the targets in the browser, it will be easy to see how the position of the robot changes.
4. To turn the function off, click the command again.

14.45. View Tool at Target

Viewing a tool at a target

1. Click **View Tool at Target** and select the tool you want to view at the target.
2. Select a target, either in the **Paths&Targets** browser or in the graphics window. You can also multiselect targets to show several copies of the tool.

   A copy of the tool will be shown at the selected target. By stepping through the targets in the browser, it is easy to see how the tool orientation changes.
3. To turn the function off, click the command and clear the check box.

P

R

S

T

U

V

W