OPAL-RT TECHNOLOGIES

RT-LAB

Quick Start Guide

Thank you for choosing RT-LAB as your real-time simulation platform.

This ***Quick Start Guide*** will guide you through the first steps in achieving real-time and closed-loop simulation, while providing explanations to get you started.

SYMBOL DEFINITIONS

The following table lists the symbols used in this document to denote certain conditions:

Symbol	Definition
	ATTENTION: Identifies information that requires special consideration
	TIP: Identifies advice or hints for the user, often in terms of performing a task
	REFERENCE _ INTERNAL: Identifies an additional source of information within the bookset.
CAUTION	Indicates a situation which, if not avoided, may result in equipment or work (data) on the system being damaged or lost, or may result in the inability to properly operate the process.
	Indicates a situation where users must observe precautions for handling electrostatic sensitive devices.
	CAUTION: Indicates a potentially hazardous situation which, if not avoided, may result in minor or moderate injury. It may also be used to alert against unsafe practices.
	WARNING: Indicates a potentially hazardous situation which, if not avoided, could result in serious injury or death.

CONTENTS

RT-LAB QUICKSTART GUIDESOFTWARE 7
SOFTWARE REQUIREMENTS 7
MATLAB™ AND SIMULINK TOOLBOXES 7
INSTALLING RT-LAB ON YOUR WINDOWS COMPUTER 8
Validating the Installation 9
STARTING RT-LAB 10
CONNECTING YOUR REAL-TIME SIMULATOR 11
Toolbar 11
CREATE YOUR FIRST PROJECT 13
USING YOUR INTEGRATION MODEL WITH I/OS 19
TROUBLESHOOTING 25
RT-LAB is not available in MATLAB™ 25
My simulator is not detected by RT-LAB 25
I need a license for my simulator 25
CONTACT AND SUPPORT 26
Resources 26

RT-LAB QUICKSTART GUIDE

SOFTWARE REQUIREMENTS

Before proceeding to install RT-LAB, make sure that you have all the required software, with appropriate versions, to ensure trouble-free installation and use. Please consult the RT-LAB Installation Guide for detailed requirements.

MATLAB™ AND SIMULINK TOOLBOXES

Before you begin your RT-LAB installation, make sure that MATLAB is installed, with the following MATHWORKS toolboxes :

1. MATLAB™ with the following toolboxes from MathWorks (see the MATLAB Compatibility tables in the Installation Guide to verify what version of MATLAB is compatible with your RT-LAB):
   - Simulink
   - Simulink Coder
   - SimPowerSystems (only for electrical simulations)

INSTALLING RT-LAB ON YOUR WINDOWS COMPUTER

If RT-LAB is not already installed on your computer, use the installation files provided on the DVD included with your delivery:

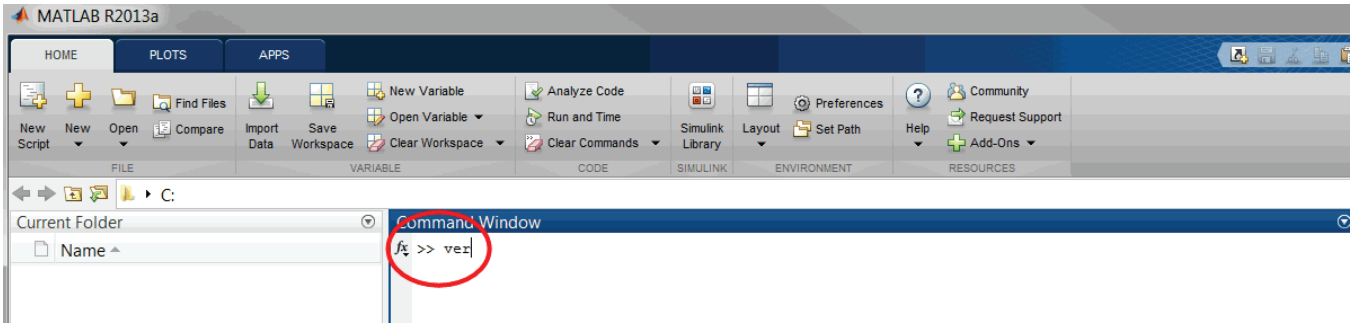
1. Insert the DVD into your computer's reader.
2. Wait for the Setup Wizard to open. If the Wizard fails to start automatically, browse the contents of the disk and run Setup/Setup. exe. (You will need administrator privileges.)
3. Follow the on-screen instructions to go through the installation process. There are several screens (up to 12) in the Wizard installation process, each one requires an interaction.
4. Please wait while RT-LAB is installed. This will only take a few minutes.
5. Once RT-LAB has been installed on your computer, click Finish to close the Wizard.

Figure 1: Setup Wizard

Validating the Installation

For validation, we recommend that you verify that all toolboxes have been installed in MATLAB™.

1. Open MATLAB™.
2. In the main page Command Window, type ‘’ver” in the command line.



3. The list of installed OPAL-RT toolboxes should include the following (only RT-LAB is mandatory. Other toolboxes may be required and depend on your licence):
   - RT-LAB
   - ARTEMIS Blockset (optional)
   - eFPGAsim (optional)
   - RT-EVENTS Blockset (optional)
   - RT-XSG (optional)

If any of these toolboxes are missing, refer to the troubleshooting section towards the end of this document to install them manually.

STARTING RT-LAB

Double-click the RT-LAB shortcut on your Desktop to launch the RT-LAB interface. The Workspace Launcher window prompts you to select a workspace (the workspace is the directory where RT-LAB will store all the files required for your simulations). We recommend you create a new and empty directory, as this will help you understand the exercises in this guide.

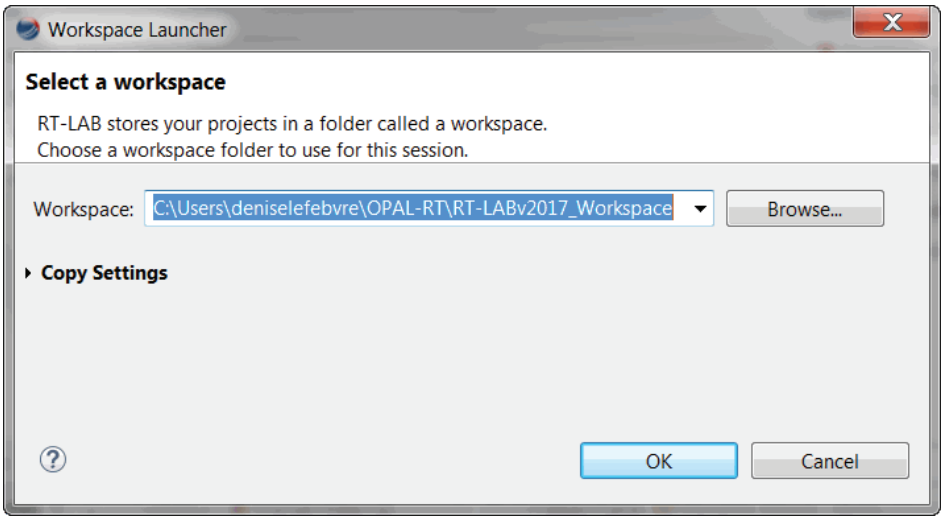


Figure 2: Selecting a workspace

Click the OK button and wait for the Welcome page to appear.

The Welcome page provides quick access to tutorials and documentation. Click **Go to the workbench** to open the main RT-LAB window (you can access the Welcome page at any time from the Help menu.)

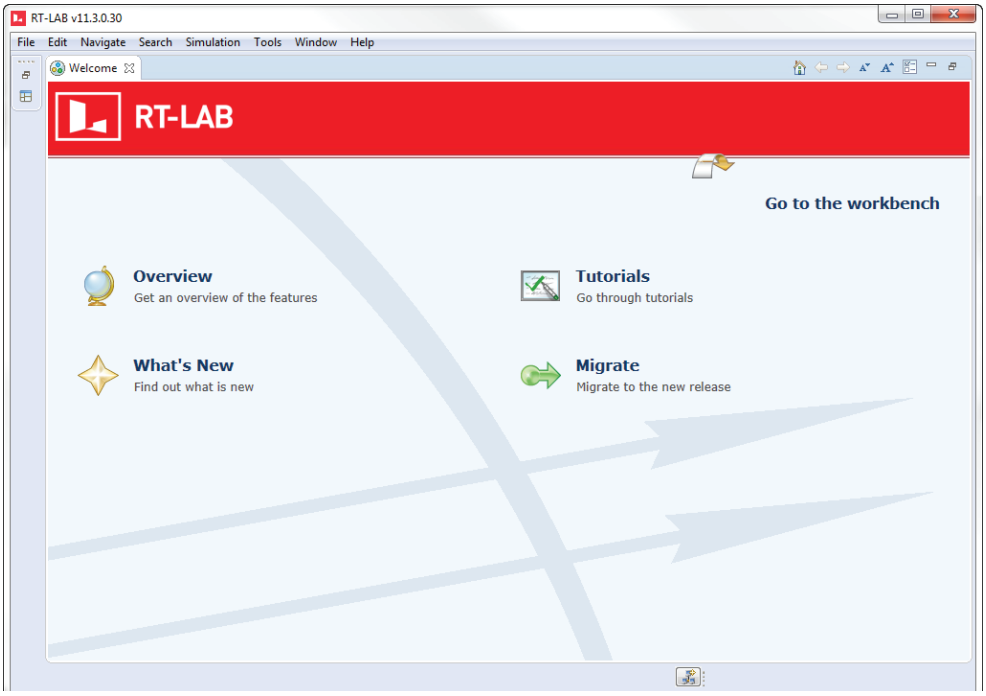


Figure 3: Welcome Page

RT-LAB is now ready to create and run real-time simulations.

CONNECTING YOUR REAL-TIME SIMULATOR

It is now time to unpack your real-time simulator (also called “target” in this document).

For the first connection, to configure your target, we recommend that you have your IT department set up your simulator’s IP address and ensure that the firewall will not interfere with or block RT-LAB.

Consult the Installation Guide for additional details (C:\OPAL-RT\RT-LAB\versionxx.x\help\pdf\RT-LAB_IG.pdf (example assumes that RT-LAB was installed on the C drive of your computer).

USING RT-LAB

Before using RT-LAB, you must configure the target. This section describes the basic tools and steps to setup RT-LAB and run a sample model.

Toolbar

In addition to the standard menu items, RT-LAB provides a toolbar of quick access buttons to do many of the tasks in one click:

Button	Name	Description
	**Build**	Build (compile) a model or manages build configurations.
	**Assign**	Opens the Assignation page of a model editor.
	**Load**	Loads a model.
	**Execute**	Starts the execution of a model.
	**Pause**	Pauses the execution of a model.
	**Reset**	Stops the execution of a model.

In the Project Explorer, double-click “Double-click to discover new targets”; this process may take some time. Once RT-LAB detects targets, the **Detected RT-LAB Targets** window appears. Select the target you want to use and click Finish

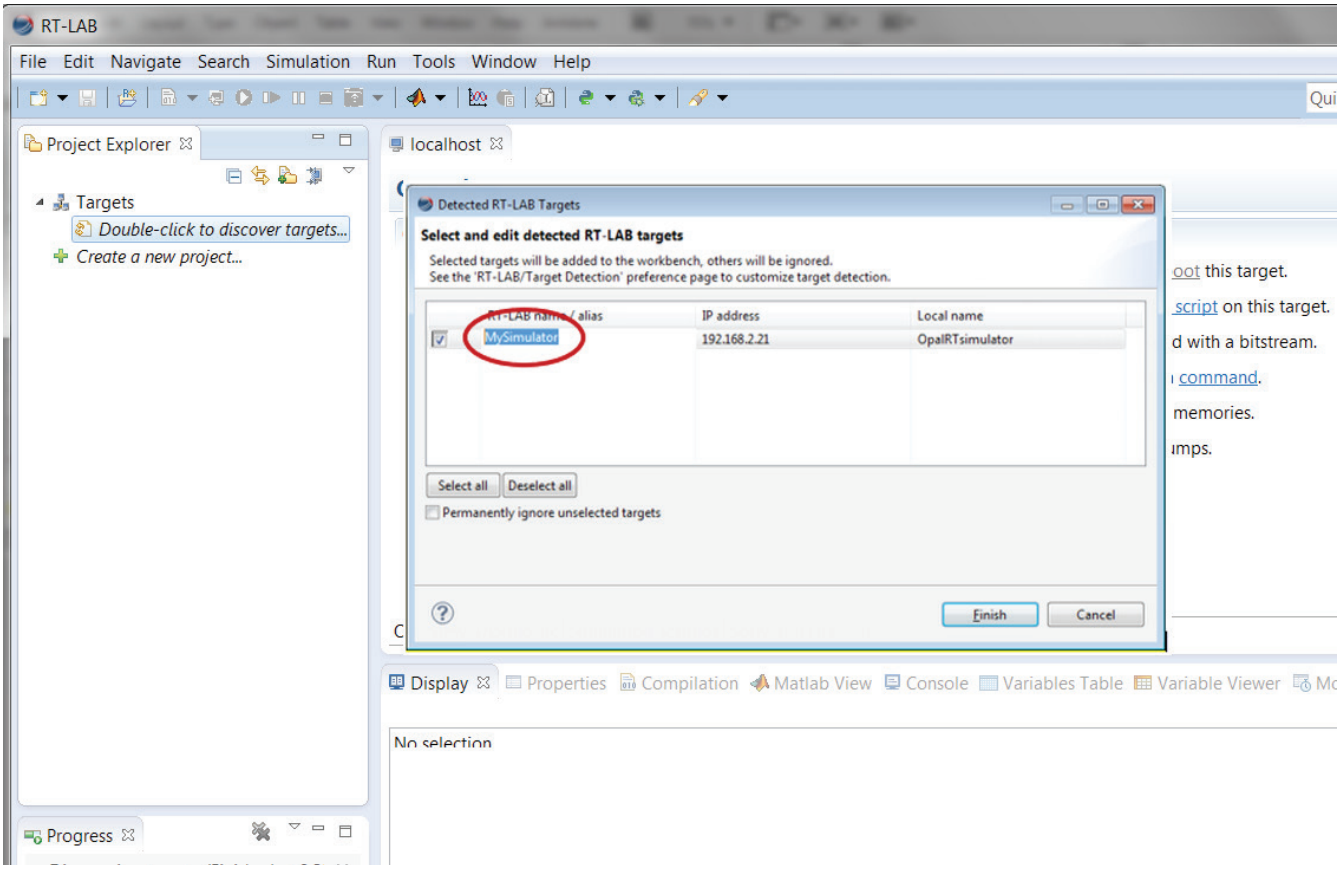


Figure 4: Discovering targets

Edit the name assigned to your simulator (in the Overview window), as desired, and click **Finish**.

Note: if your simulator is not automatically detected, please refer to the “My simulator is not detected by RT-LAB” Troubleshooting section.

Your simulator is now available in the RT-LAB interface.

CREATE YOUR FIRST PROJECT

STEP 1. Create a new project based on an example model

1. In the RT-LAB Project Explorer, double-click “Create a new project...”
2. Name your project (e.g. “My first Project”) then click Next.

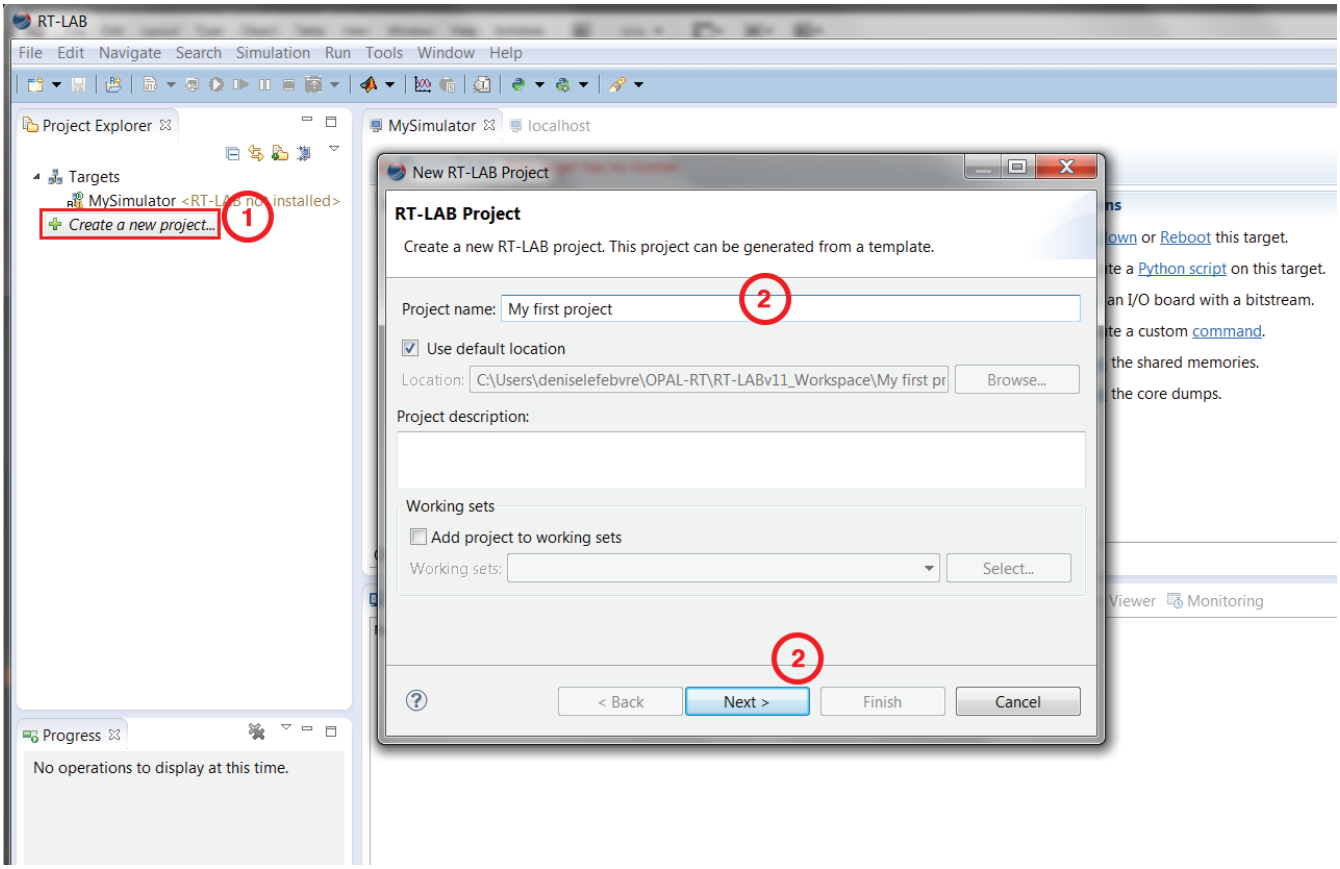


Figure 5: Creating a new project

The next window that appears allows you to select the model for the project. Select “rtdemo1” for your first project.

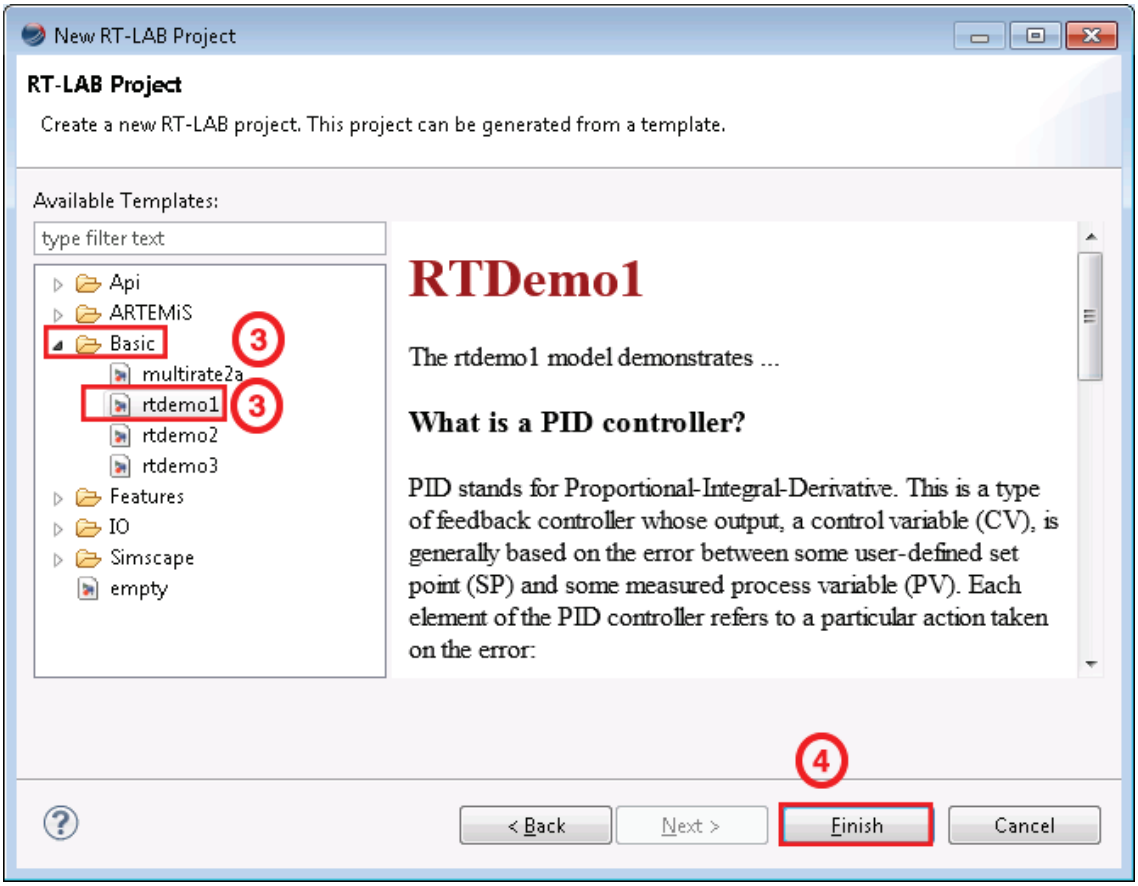


Figure 6: RT-Demo1 example

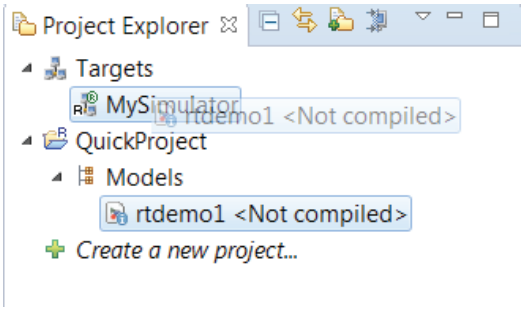
3. Browse the model directory and select the **Basic** folder and then select “rtdemo1.”
4. Click **Finish**.

Your project is now available in the Project Explorer. It contains a simple model named “rtdemo1.” This model simulates a mass-spring damper mechanical system with its PID controller.

STEP 2. Build the model

The build process allows RT-LAB to transform the Simulink model into a full real-time simulation. This process must be repeated each time the Simulink model is modified.

1. In the Project Explorer window, expand the project you just created to find the “rtdemo1” model.
2. Drag the model onto your target. This will automatically configure your model to run on this particular target.



3. Right-click on the model, then select “Simulation / Build configurations…” This opens the build configurations window

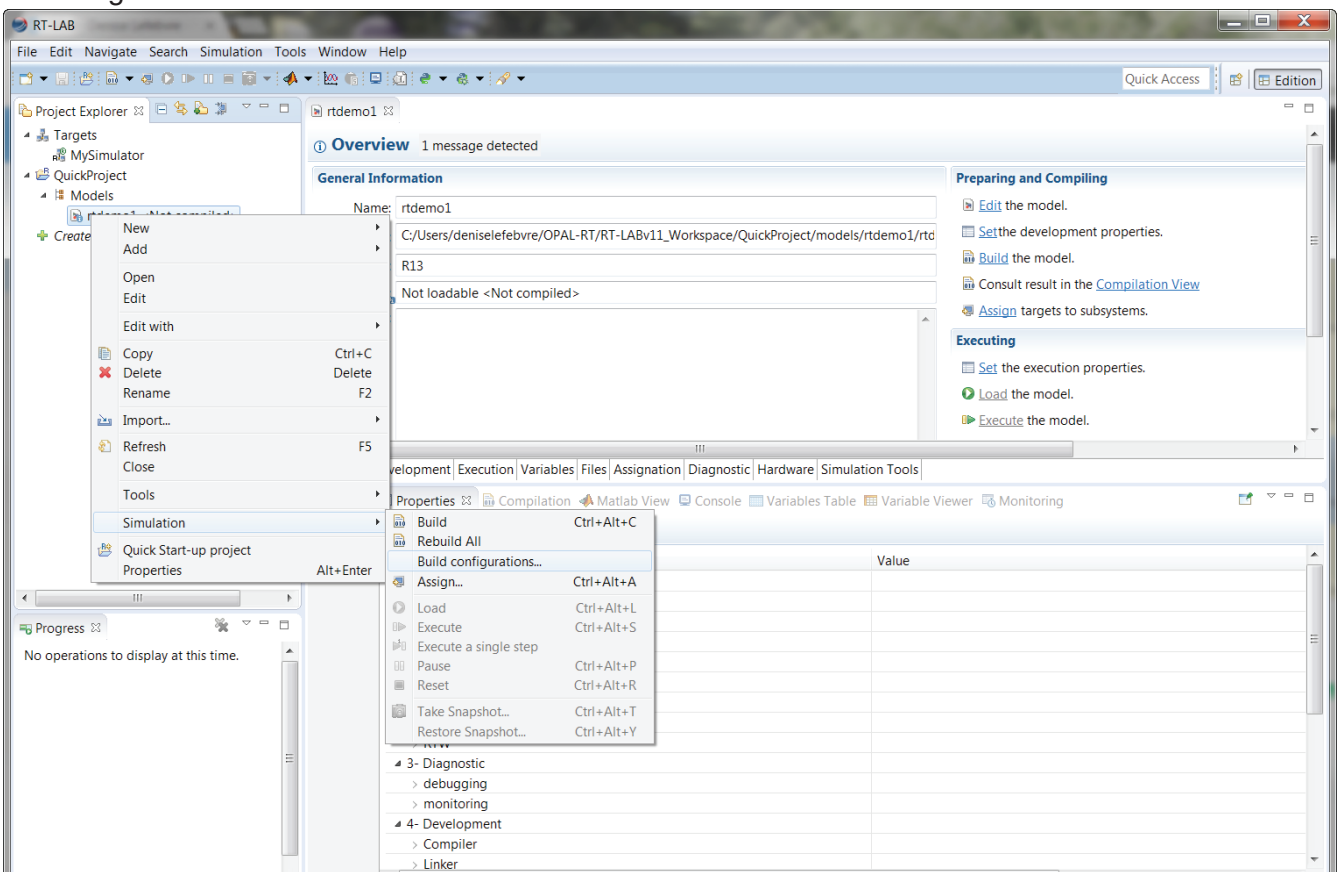


Figure 7: Building the model

4. In the Build Configurations window, select the MATLAB version to use from the drop-down list

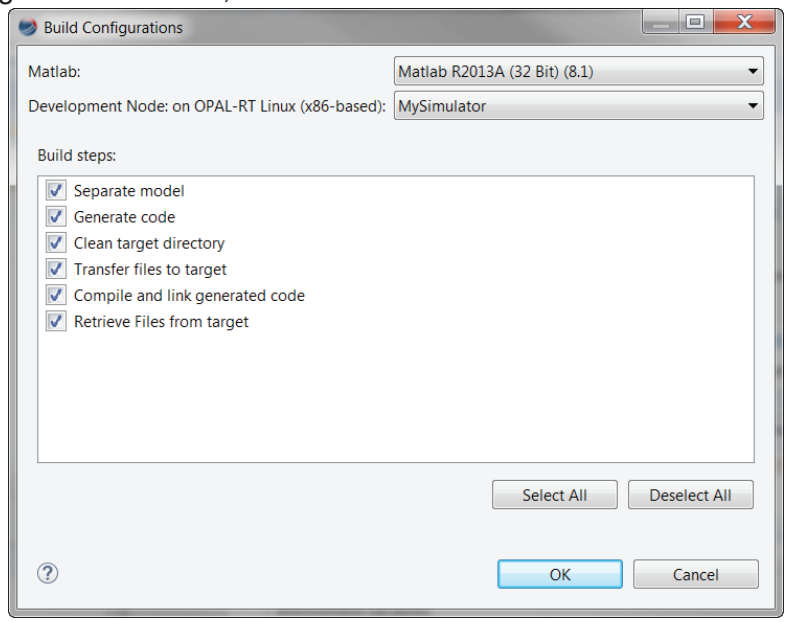


Figure 8: Building the model

5. Verify that your target is set as the “Development Node.” The “Development Node” is the target that RT-LAB will use to perform the build. (To set as Development node, right-click the target and select “Set as development node”.)
6. Click “OK” then wait for the build process to complete. You can view the progress of the build in the Compilation View at the bottom of the RT-LAB interface.

STEP 3. Load the Model

The load process prepares the real-time target to perform the simulation.

7. Click on the Load toolbar button  .

It may take a few moments for the model to load. When it has loaded, the “rtdemo1_2_sc_user_interface” Simulink console window appears (Figure 9)

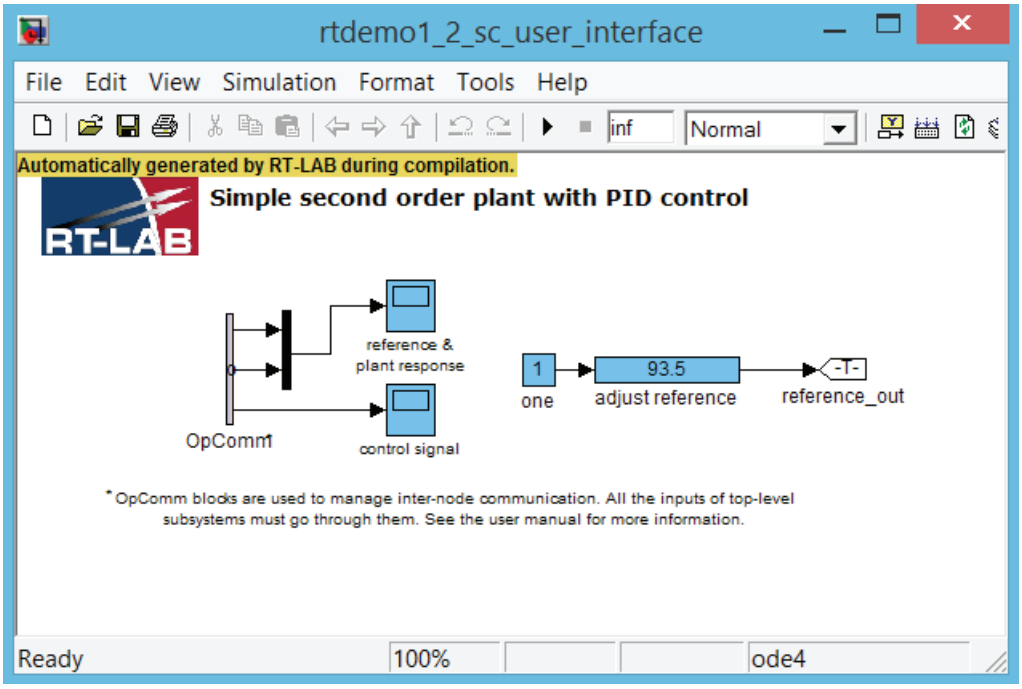


Figure 9: Simulink console

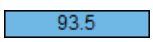
STEP 4. Execute the Model

Executing the model starts the real-time simulation on the target.

1. Click the Execute toolbar button  .

STEP 5. Use the Console to Interact With the Simulation

The user console (Figure 9) is now receiving and sending data to the simulation. Double-click a scope block  to observe the simulation and see any changes live in the scope viewer (Figure 11).

Double-click on the “adjust reference” block  to modify the set point of the mass-spring-damper system and double-click on the scope blocks to observe signals received from the simulator.

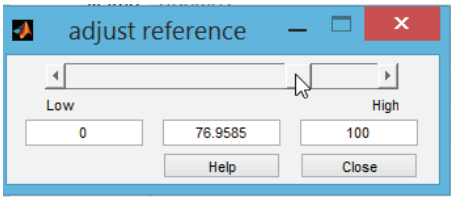


Figure 10: Adjusting the reference to view changes in scope

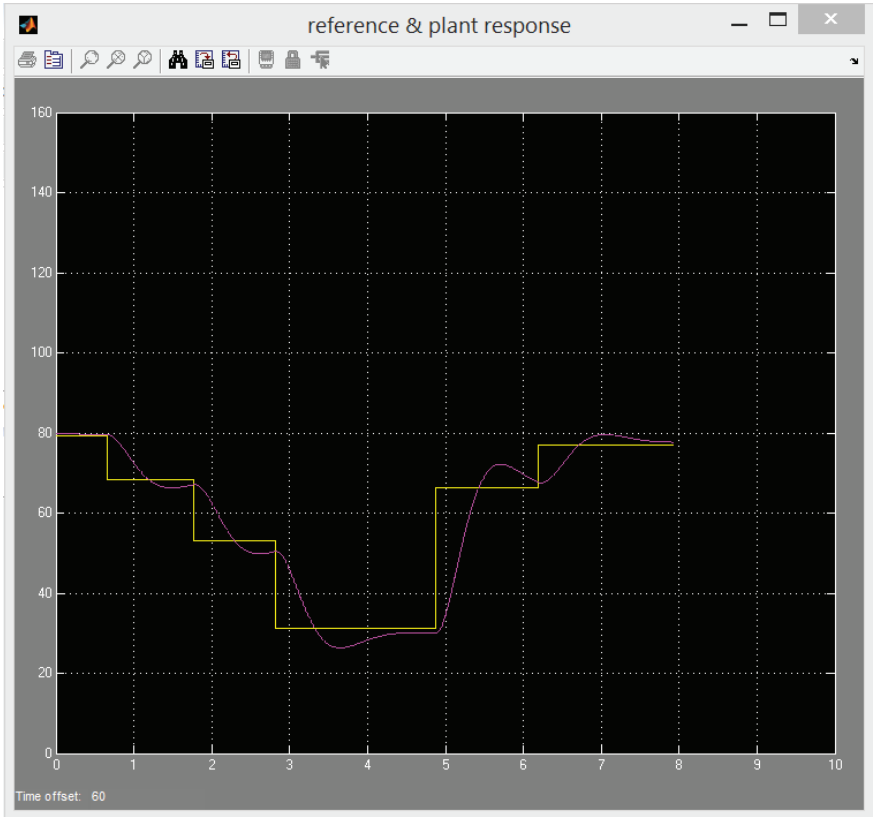


Figure 11: Observing simulation results and changes in the scope

STEP 6. Stop the Simulation

Stopping the simulation releases the target (makes it available for use) and allows for another simulation to be performed.

1. Click on the Reset toolbar button to stop the simulation .
2. Verify that the console is automatically closed. You are now ready to test your integration model.

USING YOUR INTEGRATION MODEL WITH I/OS

Depending on the system you purchased, you may have received an integration model specific to your hardware configuration; it was designed to interact with the hardware included in your simulator. Each system is delivered with a special model that uses all I/Os available on your simulator.

The model used in this guide may be different from the one you received, but the general components and instructions are similar.

STEP 1. Create a new project based on the integration model

1. Click on the *File* menu, then click ***Import***.
2. In the Import window, select RT-LAB / Existing RT-LAB Project:

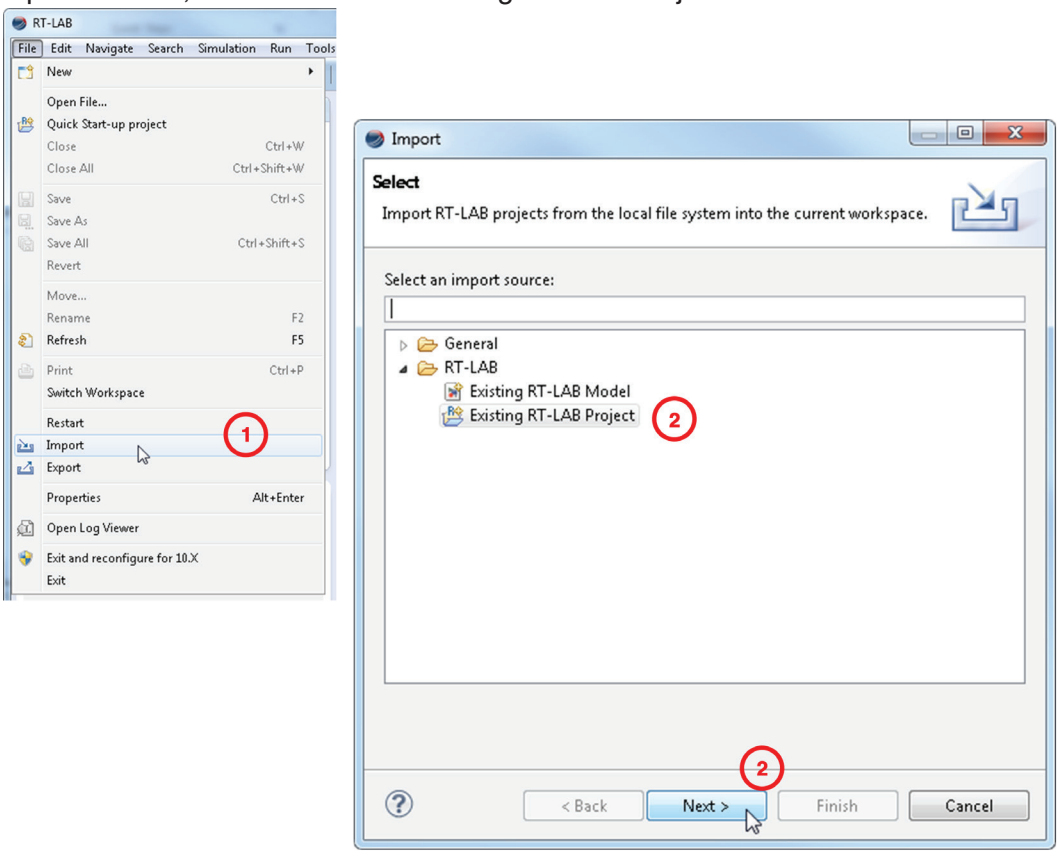


Figure 12: Importing project window

3. Click *Browse* and select the root directory of your existing project on your computer or on the DVD provided.
4. Select the project that appears in the *Projects* section.
5. Click to select the *Copy projects into workspace* option.

6. Click Finish.

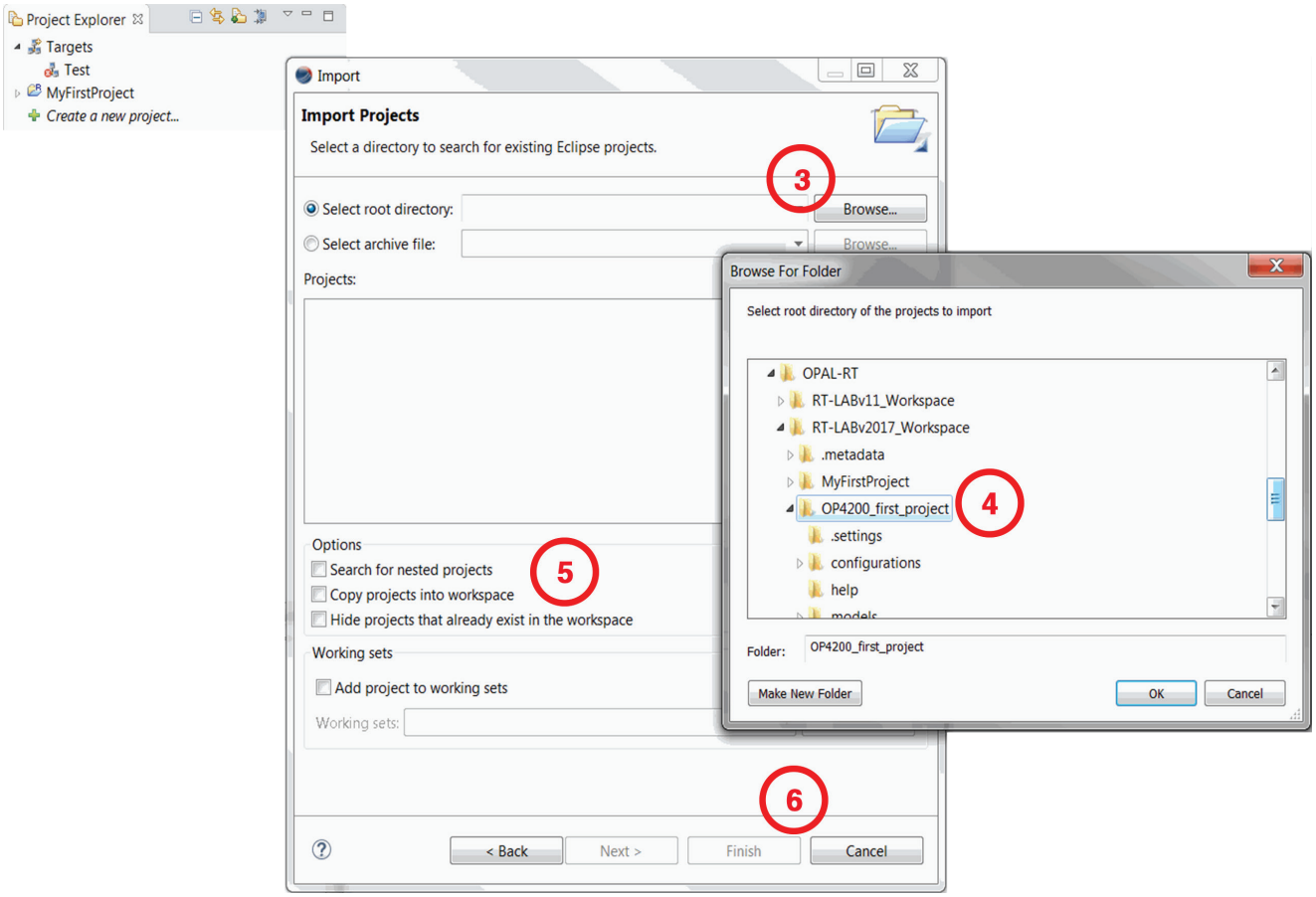


Figure 13: Import project

The project has been inported.

STEP 2. Open the model in Simulink

Right-click on the model, select *Edit with* and then click to choose your MATLAB™ version. MATLAB™ will open with your model.

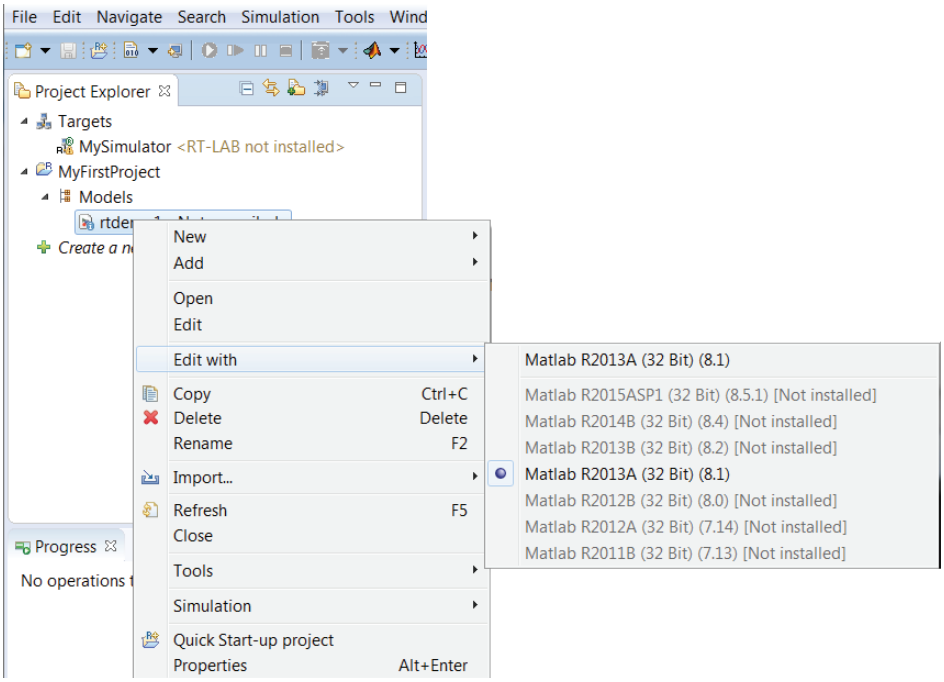


Figure 14: Editing the model

In the root layer (expand the model directory) of your Simulink model, you will find two subsystems: SM_[name] and SC_[name]:

SM_ [name] stands for subsystem master. This is where all the real- time simulation will occur. This is the only subsystem that will be run on the simulator and that contains I/Os.

The SM_[name] subsystem,a block from the OPAL-RT library that controls I/Os. Depending on your system, you may see the following blocks:

- Analog input (Ain) and output blocks (Aout),
- Static digital input (Din) and output blocks (Dout),
- PWM input (PWMin) and output blocks (PWMout), and,
- Event detector (TSDin) and Event generator blocks (TSDout).

SC_[name] stands for subsystem console. This will be an asynchronous subsystem that will run on your host computer and will act as a user interface. No critical mathematical logic should be included in this subsystem.

Note that some components of the model will be in both SM (computation) and SC (interface) subsystems (*computation components are* **only** *in the SM subsystem*).

Each of the simulator I/O configurations is described in “SystemDescription_[Customer Name]_[Project Number].pdf” included in the DVD. The I/O model will read and simulate all of your system’s I/Os.

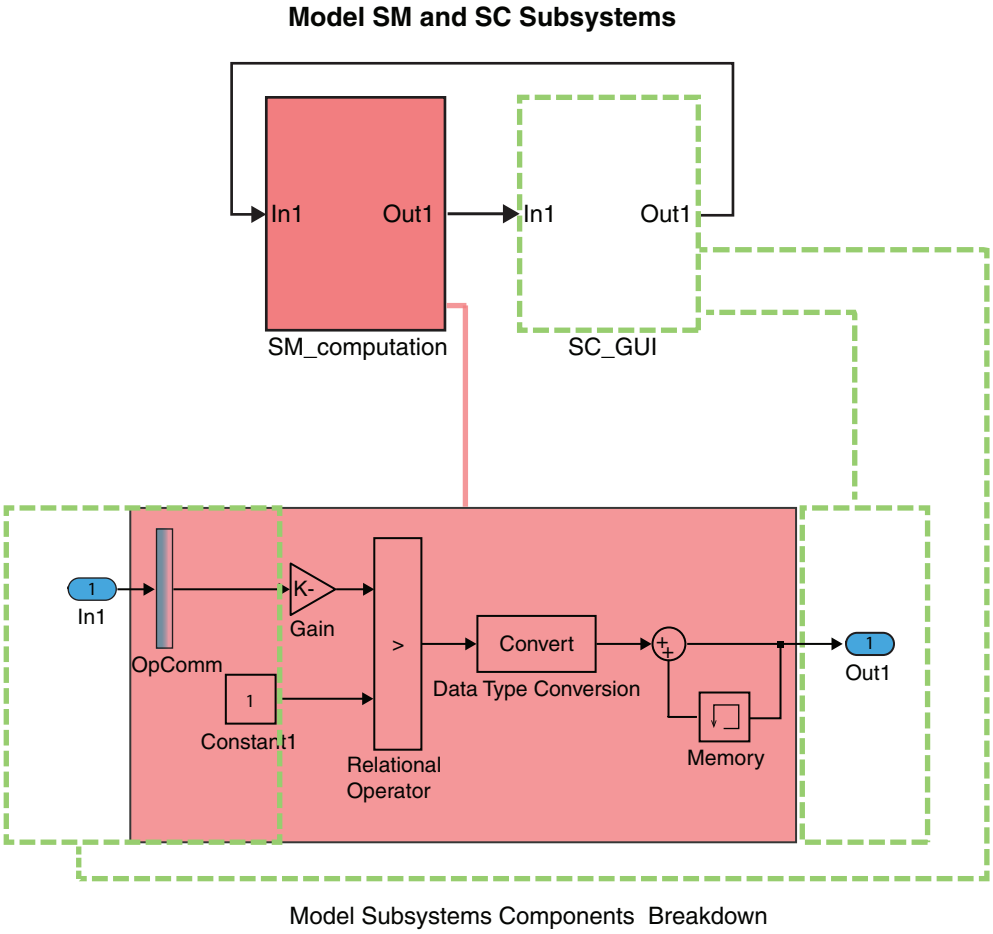


Figure 15: Example of a subsystem's components

STEP 3. Build the model

1. Drag the integration model onto your target to preconfigure it.
2. Since you have already configured the build process, simply click the Build toolbar button and wait a few seconds while the model is compiled.

STEP 4. Verify the configuration

Since this model uses I/Os, some additional steps are required.

1. Go to the **Assignation** tab and ensure that the *XHP* box is checked (on).

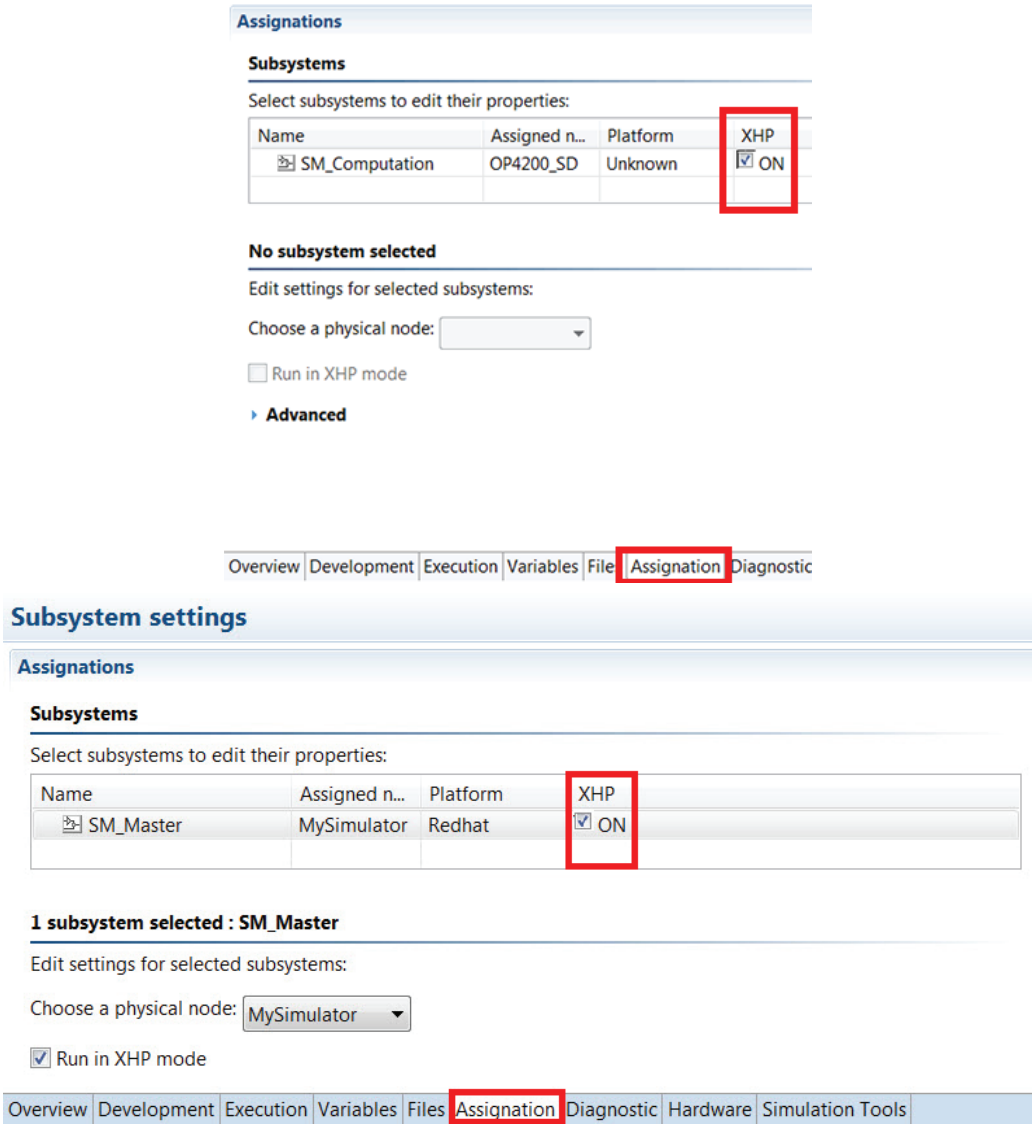


Figure 16: XHP mode

STEP 5. Load and execute the model

1. Click the Load toolbar button and wait for the load process to be completed. Click on the Execute button. A new console window appears. Your I/Os are up and running! You can now change constant blocks to control analog and digital output signals and observe analog and digital input signals using the various scope blocks Figure 17):

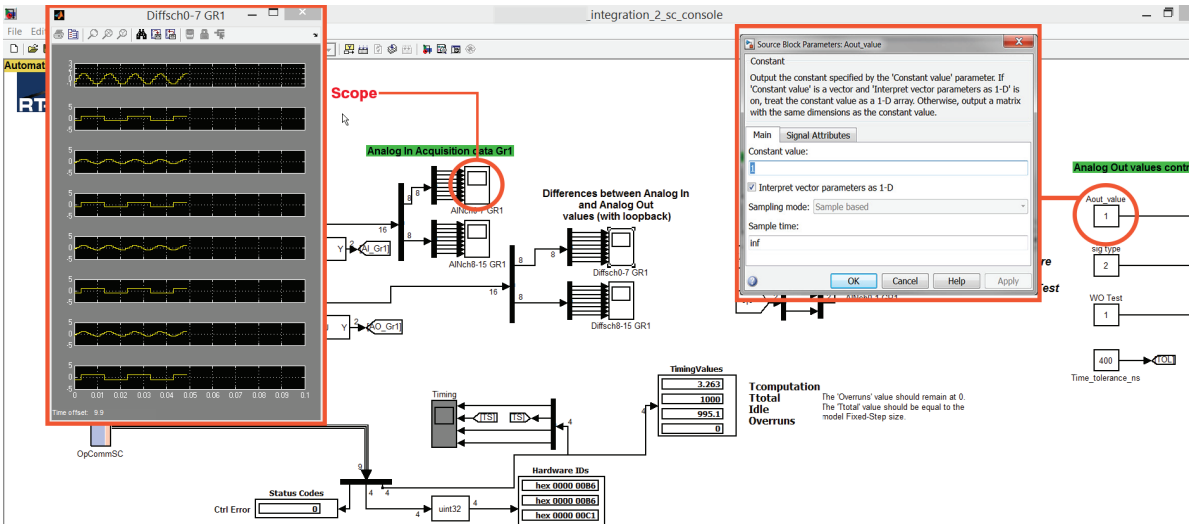


Figure 17: Sample of possible interactions with the running model

STEP 6. See external I/Os

The specific I/O configurations for your simulator are provided in the “SystemDescription_[Customer Name]_[Project Number].pdf,” document of your Integration Binder (in section B – “Mapping I/O Blocks to signal conditioning”). This is also, and most importantly, where you will find the pin assignments of each **I/O channel.**

To see external signals, you can use an oscilloscope to probe analog and digital output of the simulator. The integration model is already simulating all output signals with either a square or a sine wave signal. Note that the digital output board must be powered by an external source between 5V and 32V on the Vuser and have a ground in Vrtn pins. See the System Description documents to locate those pins.

TROUBLESHOOTING

RT-LAB is not available in MATLAB™

If any of the OPAL-RT toolboxes have not been installed in your instance of MATLAB™, you can add them manually. Run the following “m” script in your MATLAB™ to install them.

- ARTEMIS Blockset: C:\OPAL-RT\ARTEMIS\[ARTEMIS version]\art_m\setup_artemis.m
- RT-LAB: C:\OPAL-RT\RT-LAB\[RT-Lab version]\simulink\m\setup_rtlab.m
- RT-XSG: C:\OPAL-RT\RT-XSG\[RT-XSG version]\Simulink\xsg_Rxx\m\setxsgpath.m

My simulator is not detected by RT-LAB

1. Make sure that your antivirus or firewall software has not blocked RT-LAB.
2. Create a target node in RT-LAB: in the Project Explorer, right-click on Targets then select New / New Target.
3. Enter a name and the specific IP address for the target.
4. Now click Ping to ensure that the target is available. If not, please contact your network administrator or follow the instructions on this page: http://www.opal-rt.com/kb-article/how-change-ip-address-or-ip-mask-redhat-target-through-telnet. Otherwise, simply click Finish and your target will appear in the Project Explorer.

I need a license for my simulator

If your simulator needs a license, double-click on it in the Project Explorer to open its editor, then go to the License tab and follow the instructions.

For more details on the license system, please visit: http://www.opal-rt.com/KMP/index.php?/article/AA-01022/8/HowTo/How-To-Request-and-Install-a-License-for-RT-LAB-11.x.html

CONTACT AND SUPPORT

If you have any questions, please refer to our Resource Center or our Download Center web pages or contact the Support team using the “Support Request” web page: www.opal-rt.com/support-home.

Be sure to check out the Troubleshooting page of this Quick Start Guide. It is a good reference for helping you understand the source of any issues you might have. If you don’t find an answer, please contact us and we will answer your questions and help you with the getting started process.

Resources

Download latest version	http://www.opal-rt.com/download-center/
Technical Support	www.opal-rt.com/support-home
Knowledge Base	www.opal-rt.com/support-knowledge-base

CONTACT

OPAL-RT Corporate Headquarters

1751 Richardson, Suite 2525
Montréal, Québec, Canada
H3K 1G6
Tel.: 514-935-2323
Toll free: 1-877-935-2323

Technical Services
www.opal-rt.com/support

OPAL-RT Technologies Inc.

QS16-47318-RVN 1.0
07/2020