

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
8	ATTENTION: Identifies information that requires special consideration
	TIP: Identifies advice or hints for the user, often in terms of performing a task
6	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
Тооlbar	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.

MATLAB R.	2013a									
HOME	PLOTS	APPS								
New New O Script 👻	Find Files	Import Save Data Workspace	New Variable Open Variable Clear Workspace	Analyze Code	Simulink Library	Layout	 Preferences Set Path 	? Help	 ➢ Community ➢ Request Support ☆ Add-Ons ▼ 	
F	ILE	V	ARIABLE	CODE	SIMULINK	E	NVIRONMENT		RESOURCES	
	I → C:									
Current Folde	r		Command Wind	dow						C
🗋 Name 🕯			fr >> ver							

- 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
010	Build	Build (compile) a model or manages build configurations.
Q	Assign	Opens the Assignation page of a model editor.
0	Load	Loads a model.
	Execute	Starts the execution of a model.
00	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





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



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.

SRT-LAB	maxing some at our Dr Mr Rr	
File Edit Navigate Search Simulation Run	Tools Window Help	
😂 🕶 🔛 📇 🔠 🖛 🗟 🜔 💷 🔳 🗃 🕶 4	A 🕶 🗠 👘 🙆 👻 🖛 🍕 🕶 🖋 🕶	
Project Explorer 🛛 🖳 🗖	Iccalhost	
😑 😓 🎘 🎽	New RT-LAB Project	
R [®] MySimulator <rt-l 6="" installed="" no=""> H Create a new project</rt-l>	RT-LAB Project	ns
	Create a new RT-LAB project. This project can be generated from a template.	own or <u>Reboot</u> this target.
		an I/O board with a bitstream
	Project name: My first project	te a custom command.
	Use default location	the shared memories.
	Location: C:\Users\deniselefebvre\OPAL-RT\RT-LABv11_Workspace\My first pr Browse	the core dumps.
	Project description:	
	Working sets	
	Add project to working sets	
	Working sets: Select	
	d	Viewer 🐻 Monitoring
	Cancel	
To Progress 🛛 🦋 🍟 🗖		
No operations to display at this time.		

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

RT-LAB		a present of the							
File Edit I	Vavig	gate Search Simulation	Tool	s Window H	lelp				
🗅 🕶 🔛 d	9 6) - @ 0 ⊳ п = 🖬	- 📣	- 120 👘 🗳		e • e • 🛷 •			Quick Access 🔡 🖽 Edition
Project E	kploi	rer 🛿 🖻 🛸 🔈 🍸 🖤		🖹 rtdemo1 🕯	23				
🔺 💑 Targe	ts			n Overvi	ew	1 message detected			
📲 My	Simu	ulator							
	dels			General Inf	orm	ation			Preparing and Compiling
				Name	e: rto	demo1			Edit the model.
+ Create		New		*	C;	/Users/deniselefebvre/	OPAL-RT/RT-LABv	11_Workspace/QuickProject/models/rtdemo1/rtc	Set the development properties.
		Add		,	R	13			Build the model.
		Open Edit				ot loadable <not com<="" td=""><td>oiled></td><td></td><td>Consult result in the Compilation View</td></not>	oiled>		Consult result in the Compilation View
		Eult			ľ			*	Assign targets to subsystems.
		Edit with		•					Executing
		Copy		Ctrl+C Delete					Set the execution properties.
		Rename		F2					Load the model.
	2	Import		•					Execute the model.
	ଛା	Refresh		F5	L			m	
		Close			velo	pment Execution Varia	ables Files Assign	ation Diagnostic Hardware Simulation Tools	
		Tools		۱.	Pro	perties 🛛 🗟 Compila	ation 📣 Matlab Vi	ew 🖾 Console 🥅 Variables Table 🎹 Variable	Viewer 🖬 Monitoring 📰 🗖 🗖
		Simulation		۰,	60	Build	Ctrl+Alt+C		
	1 ^{BQ}	Ouick Start-up project			.	Rebuild All			
		Properties		Alt+Enter	_	Build configurations Value			
•	_	III	Þ		Q.	Assign	Ctri+Alt+A		
Program	~	× v				Load	Ctrl+Alt+L		
- Fridgress	~				pie -	Execute a single step	Carratos		=
No operati	ons t	to display at this time.				Pause	Ctrl+Alt+P		
						Reset	Ctrl+Alt+R		
					6	Take Snapshot	Ctrl+Alt+T		
						Restore Snapshot	Ctrl+Alt+Y]	
			Ξ		⊿ 3	- Diagnostic			
						debugging			
					4.4	> monitoring			
					= 4	> Compiler			
						> Linker			

Figure 7: Building the model

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

Build Configurations	
Matlab:	Matlab R2013A (32 Bit) (8.1)
Development Node: on OPAL-RT Linux (x86-based):	MySimulator •
Build steps:	
📝 Separate model	
Generate code	
Clean target directory	
Transfer files to target	
Compile and link generated code	
Retrieve Files from target	
	Select All Deselect All
	OK Cancel

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 💟.

Ô

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)

📓 rtdemo1_2_sc_user_interface — 🗖 🗙
File Edit View Simulation Format Tools Help
D 🖆 🖬 🚭 ※ 🖻 💼 수 수 수 으 으 ▶ = inf 🛛 Normal 💽 🛱 🛗 🔮 🤅
Automatically generated by RT-LAB during compilation. Simple second order plant with PID control
Ready 100% ode4

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 <u>93.5</u> to modify the set point of the mass-springdamper system and double-click on the scope blocks to observe signals received from the simulator.

🛃 adjust re	eference	_ □	×
۲ Low		-2	▶ High
0	76.9585		100
	Help	(Close

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 \blacksquare .
- 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.

RT-LAB Quickstart Guide Using Your Integration Model With I/Os

6. Click Finish.			
Project Explorer 🛛 🕒 🕏 🖡	≥ " □ □		
 ▲ Saragets ▲ Test 	Import		
 MyFirstProject Create a new project 	Import Projects Select a directory to search for existing Eclipse projects.		
	Select root directory:	Browse	
	Select archive file:	Browse B	
		Select root directory of the projects to import	
		OPAL-RT DAL-RT D RT-LABv11_Workspace	A
		RT-LABv2017_Workspace	
		Interducta Image: Inter	E
	Options	L .settings	
	Search for nested projects	help	
	Hide projects that already exist in the workspace	models	+
	Working sets	Folder: OP4200_first_project	
	Add project to working sets	Make New Folder	OK Cancel
	Working sets:		
		6	
	ABack Next >	Finish Cancel	

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.

File Edit Navio	gate	Search	Simulation	Tools	Wind		
📬 👻 🖩 😫 🖡	•	a O i	▶ II = []	- 📣	- 12	-	
Project Explor	rer 8	3	8 🕸 🔈	~ -			
🔺 💑 Targets							
📲 MySimu	lato	r <rt-la< td=""><td>B not installed</td><td>></td><td></td><td></td><td></td></rt-la<>	B not installed	>			
▲ I Models	ject						
R rtder	4	NI 1					
🕂 Create a n		New				•	
		Add				•	
		Open					
		Edit					
		Edit with	n			•	Matlab R2013A (32 Bit) (8.1)
		Сору			0	Ctrl+C	Matlab R2015ASP1 (32 Bit) (8.5.1) [Not installed]
	×	Delete				Delete	Matlab R2014B (32 Bit) (8.4) [Not installed]
		Nename	;			FZ	Matlab R2013B (32 Bit) (8.2) [Not installed]
	2	Import				•	Matlab R2012R (32 Bit) (8.1) Matlab R2012R (32 Bit) (8.0) [Not installed]
	\$1	Refresh F5					Matlab R2012A (32 Bit) (7.14) [Not installed]
= Progress 🛛		Close					Matlab R2011B (32 Bit) (7.13) [Not installed]
No operations t		Tools				+	
		Simulati	ion			+	
	ピ	Quick St Properti	art-up projec es	t	Alt+	Enter	

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).

	Calanta la desta	that the second					
	Select subsystems to e	dit their propert	ties:				
	Name	Assign	ed n Pl	atform	XHP		
	B SM_Computation	on OP4200	0_SD U	hknown	I⊻ OI	N	
	No subsystem select	ed					
	Edit settings for select	ed subsystems:					
	Choose a physical nod	le:	-				
	Run in XHP mode						
	Advanced						
	Overview Development	Execution Varia	ables File	Assignati	on Diagn	ostic	
Ibsystem setti	Overview Development	Execution Varia	ables File	Assignati	on <mark>Diagn</mark>	ostic	
Ibsystem setti ssignations Subsystems	Overview Development	Execution Varia	ables File	Assignati	on <mark>Diagn</mark>	ostic	
Ibsystem setti ssignations Subsystems Select subsystems to	Overview Development ngs edit their properties:	Execution Varia	ables File	Assignati	on Diagn	ostic	
Ibsystem setti ssignations Subsystems Select subsystems to Name	Overview Development ngs edit their properties: Assigned n	Execution Varia	Ables File	Assignati	on <mark>Diagn</mark>	ostic	
Ibsystem setti ssignations Subsystems Select subsystems to Name Mame	Overview Development ngs edit their properties: Assigned n MySimulator	Execution Varia	XHP	Assignati	on <mark>Diagn</mark>	ostic	
Ibsystem setti ssignations Subsystems Select subsystems to Name	Overview Development ngs edit their properties: Assigned n MySimulator ed : SM_Master cted subsystems:	Execution Varia	XHP	Assignati	on Diagn	ostic	

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: <u>www.opal-rt.com/support-home</u>.

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 Note:

While every effort has been made to ensure accuracy in this publication, no responsibility can be accepted for errors or omissions. Data may change, as well as legislation, and you are strongly advised to obtain copies of the most recently issued regulations, standards, and guidelines.

This publication is not intended to form the basis of a contract.



OPAL-RT Technologies Inc.

QS16-47318-RVN 1.0 07/2020