

Simulink DLL based Interface

SimulinkDLL1

Simulink DLL
Interface

Simulink DLL based Interface	1
1 Description	1
2 Requirements and Compatibility	1
3 DLL generation and integration steps	2
3.1 Initial setup in MATLAB	2
3.2 Steps for creating a DLL	2
3.2.1 Sample Simulink diagram	3
3.2.2 Code generation steps	4
3.2.3 Importing the model into EMTWorks	8
3.2.4 Rules and limitations	10
3.2.5 Searching for tunable parameter problems on Simulink side	10
4 Simulink model options and configurations	12
4.1 Model Calling Rule	13
4.1.1 Simulink time-step (default option)	13
4.1.2 User-defined	13
4.1.3 On EMT time-mesh	13
4.1.4 At any time-step	13
4.2 Other Settings	13
4.2.1 DLL version	13
4.2.2 Interpolation and Extrapolation options	14
4.3 Behaviour when $\Delta t_{EMT} \neq \Delta t_{CallSimulink}$	14
5 Compatibility with EMT simulation options	15

César Martin (RTE-France), Jean Mahseredjian, 5/11/2023 1:37:00 AM

1 Description

This device named “Simulink DLL Interface” (SDI) is used to interface EMT with DLLs created using Simulink® (MATLAB®). The SDI is an EMT Toolbox.

Once a DLL is created using Simulink and a compiler, this device is used to create all the necessary connections (pins) and provide access to some modifiable parameters on the Simulink side.

The interfacing pins (ports) on the Simulink side must be created by the user. The interfacing pins in EMTWorks are automatically created with the SDI device. The user must then finalize all the desired connections in EMTWorks to use the Simulink code in the simulations on EMT side.

2 Requirements and Compatibility

The user must have a valid MATLAB license with Simulink. The following MATLAB Toolboxes are minimal requirements, but this may change according to MATLAB versions. Required Toolboxes:

1. Matlab Coder
2. Simulink Coder
3. Embedded Coder

Some other Toolboxes may be required according to Simulink designs.

This version of the SDI has been tested with the following MATLAB versions: R2013b to R2021b. It may also work with older Matlab versions.

It is noticed that under some circumstances, especially when S-Functions and MATLAB functions are used in Simulink diagrams, the simulation results may be corrupted or the DLL generation process may fail. It is the user's responsibility to verify coherency in simulation results.

In addition to the above MATLAB requirements, the user must install a C++ compiler. The compatible compilers are Microsoft Visual Studio C++ version 2010, 2013, 2015, 2017 and 2019. The Visual Studio C++ version must be compatible with the Matlab version used. See "supported compilers" for each Matlab version here <https://fr.mathworks.com/support/requirements/previous-releases.html>

3 DLL generation and integration steps

In the following sections, the name ApplicationDir designates the application folder (path):

1. On a 64 bit Windows system the default value is C:\Program Files (x86)\EMTPWorks.
2. On a 32 bit Windows system the default value is C:\Program Files\EMTPWorks.

The EMTP Toolboxes are located in the folder ApplicationDir\Toolboxes.

The SDI Toolbox is located in the folder ApplicationDir\Toolboxes\SimulinkDLL.

The name UserDataFolder designates the EMTP Documents folder in Windows. On a typical computer this is given by:

C:\Users\\Documents\EMTP

The name ModelName.dll designates the DLL file generated from Simulink.

3.1 Initial setup in MATLAB

The following commands must be executed in MATLAB to prepare for automatic generation of DLLs from Simulink.

1. The TLC_EMTP folder
As a first step it is necessary to access the TLC_EMTP folder. This folder is normally located in the UserDataFolder. An initial version of this folder is also found in ApplicationDir\Toolboxes\SimulinkDLL.
The user must run the following command in the MATLAB window:

```
>> addpath(genpath('UserDataFolder\SimulinkDLL\TLC_EMTP'))  
>> savepath
```
2. Selecting the compiler in MATLAB
If the C++ compiler is not already selected in MATLAB, the user must run the following MATLAB command:

```
>> mex -setup or mex -setup c++
```

3.2 Steps for creating a DLL

The Simulink diagram may contain the following:

1. One or several Input ports (Inports) for interfacing on EMTPWorks side
2. One or several Output ports (Outports) for interfacing on
3. The ports may be real or complex of dimension 1
4. The ports may be real or complex of dimension greater than 1
5. One or several Subsystems
6. Different sampling rates
7. Tunable parameters: parameters that can be modified in EMTPWorks side
8. MATLAB functions
9. S-Functions

Under some circumstances the MATLAB functions and S-Functions may not perform as expected and the user must verify that the simulation results are coherent.

3.2.1 Sample Simulink diagram

The Simulink example shown in Figure 1 is used here to demonstrate the DLL generation steps. The top level model (design) is shown in Figure 1a and the contents of the sample Subsystem are shown in Figure 1b.

The Simulink model (mdl) file corresponding to Figure 1 can be found in the folder ApplicationDir\Toolboxes\SimulinkDLL\Example\SimpleDemo.

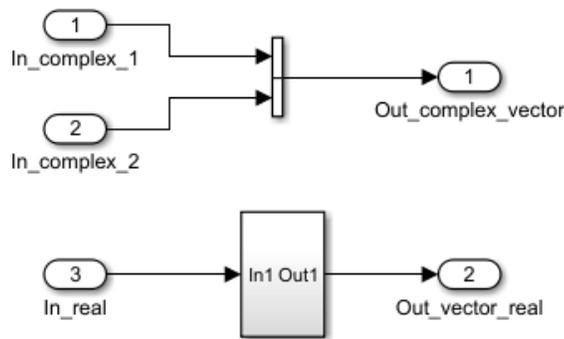
The individual components on Simulink side must be correctly specified using Simulink input data tabs. In this example:

1. The ports In_complex_1 and In_complex_2 have "Port Dimensions" set to 1 and "Signal type" set to complex. The "Sample time" is -1 for inherited.
2. For the In_real port, "Port dimensions" is set to -1, "Sample time" is set to Ts, the "Signal type" is auto.
3. The Out_complex_vector is set to inherit its "Port dimensions" and "Signal type" is auto which makes it of dimension 2 and of complex type.
4. The Out_vector_real port inherits its dimension of 2 and its "Signal type" type is also inherited as real-type.
5. For the Subsystem:
 - a. the gain block Gain_K is set to the variable value K
 - b. the gain block Gain_L is set to the variable value L
 - c. the Discrete-Time Integrator has an inherited sample time
 - d. the Discrete-Time Integrator_Tsx100 has a sample time of 100*Ts

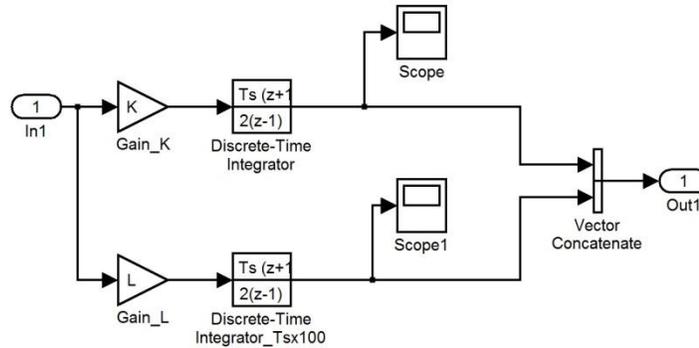
Since this example contains variables in some of its blocks, it is necessary to initialize those variables in the MATLAB work space. In this case it is necessary to enter:

```
>> K = 2;  
>> L = 3;  
>> Ts = 10e-6;
```

Or to run the provided m-file init_data.m.



a) Top level model



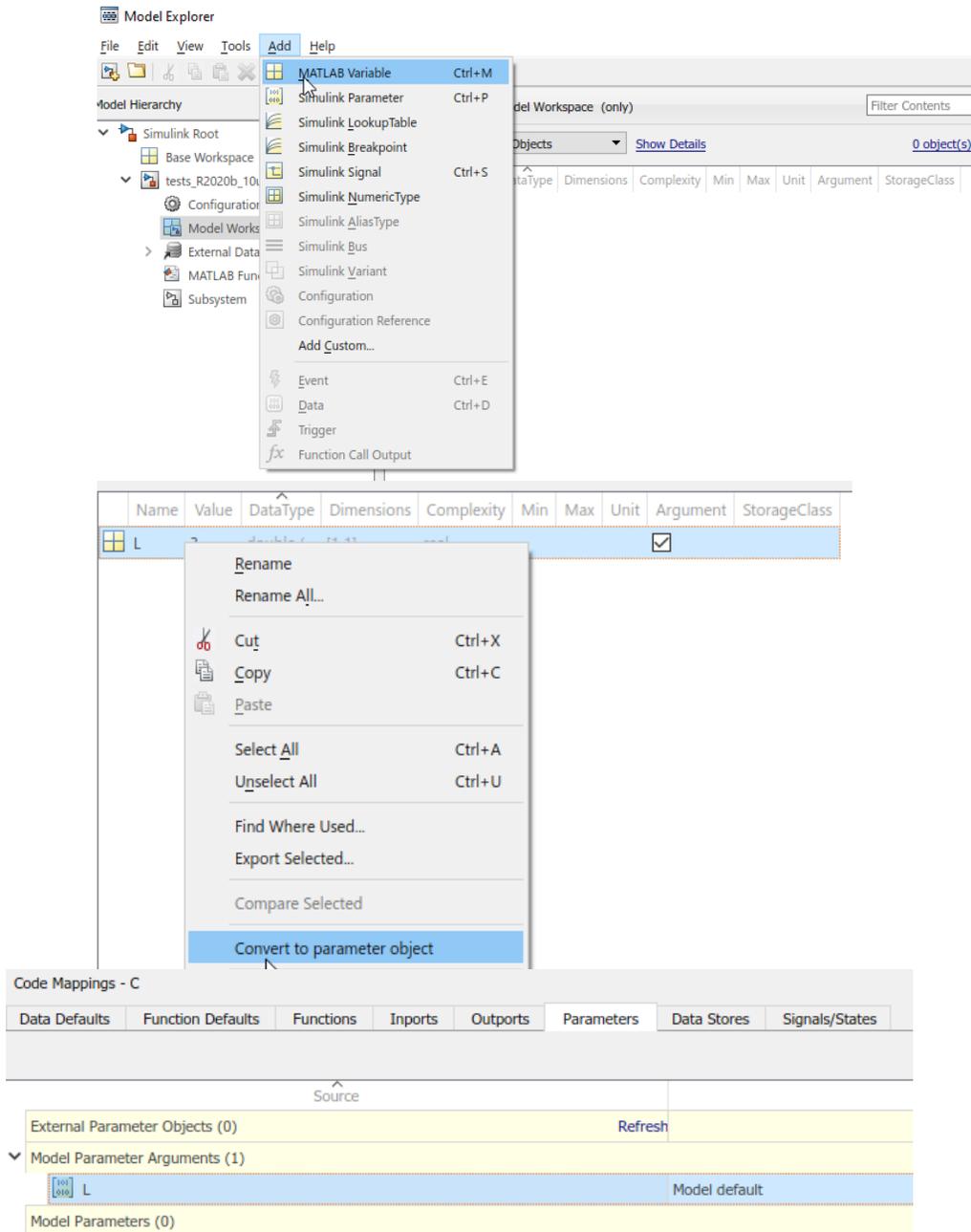
b) Subsystem

Figure 1 Sample Simulink diagram

3.2.2 Code generation steps

The steps are as follows:

1. Select the configuration parameters using the Simulink menu “Simulation>Model Configuration Parameters” (or by clicking on CTRL+E) and click on Code Generation.
 - a. It is necessary to locate and select the “System target file” EMTP.tlc using the Browse button as shown in Figure 2.
 - i. In the shown example, there is no referenced model, but it is allowed to use such models by setting exactly the same target file (EMTP.tlc) and the same configuration parameters in all references models. It is not allowed to use tunable parameters when referenced models are used.
 - b. Under the “Code Generation” submenu, select “EMTP code generation options” (see Figure 3):
 - i. Delete source files after code compilation: this option can be used to delete source code for confidentiality reasons.
2. All files related to this model (design) must be located in the same folder as the model file.
3. Select Solver (see Figure 4) and verify that the Fixed-step size is set to the variable Ts.
4. In this optional step it is possible to select tunable parameters. The tunable parameters are parameters that can be modified from EMTPWorks without recompiling (regenerating) the Simulink DLL. Two different methods depending on the Matlab version used.
 - a. For Matlab version R2018b or older:
 - i. In the menu Optimization>Signals and Parameters, click on the Configure button (when “Inline parameters” is checked).
 - ii. Select the “Source list” being “Matlab workspace variables”.
 - iii. Select the tunable parameters, and click on “Add to table”, as shown in Figure 5. The tunable parameters can be of type real, logical or integer. Complex numbers, vectors and matrices are not allowed. Some parameters (like Ts) cannot be tunable.
 - iv. Click OK to proceed.
 - b. For Matlab version R2019a or newer:
 - i. From the Simulink circuit, open « Model Explorer » under the Modeling tab or click CTRL+H
 - ii. In the Model Hierarchy on the left, select your Model workspace and click on Add then Matlab variable (or move the variable from the Matlab workspace to the Model workspace if the variable was previously created in the Matlab workspace)
 - iii. Choose a name and set a value to the parameter if it hasn't been done already
 - iv. Check the “argument” box
 - v. Right click on the variable, then “Convert to parameter object”
 - vi. Under “StorageClass” click on Configure
 - vii. Select “Model default » for the storage class



- At this stage the setup is ready to generate the DLL by clicking on the Build button in the top panel of the Simulink circuit or by clicking on CTRL+B from the Simulink circuit. If there are no error messages in MATLAB, then the build is successful and two new folders will appear in the same folder as the compiled model file folder: slprj and ModelName_EMTP_rtw. In this example the ModelName_EMTP_rtw folder is tests_R2018b_10us_full_EMTP_rtw. The final DLL file of the model is also located in the same folder as the model file. Its generic name is ModelName.dll (tests_R2018b_10us_full.dll for this example). The two folders slprj and ModelName_EMTP_rtw are not needed and can be deleted, only the DLL file is needed to run in EMTP.

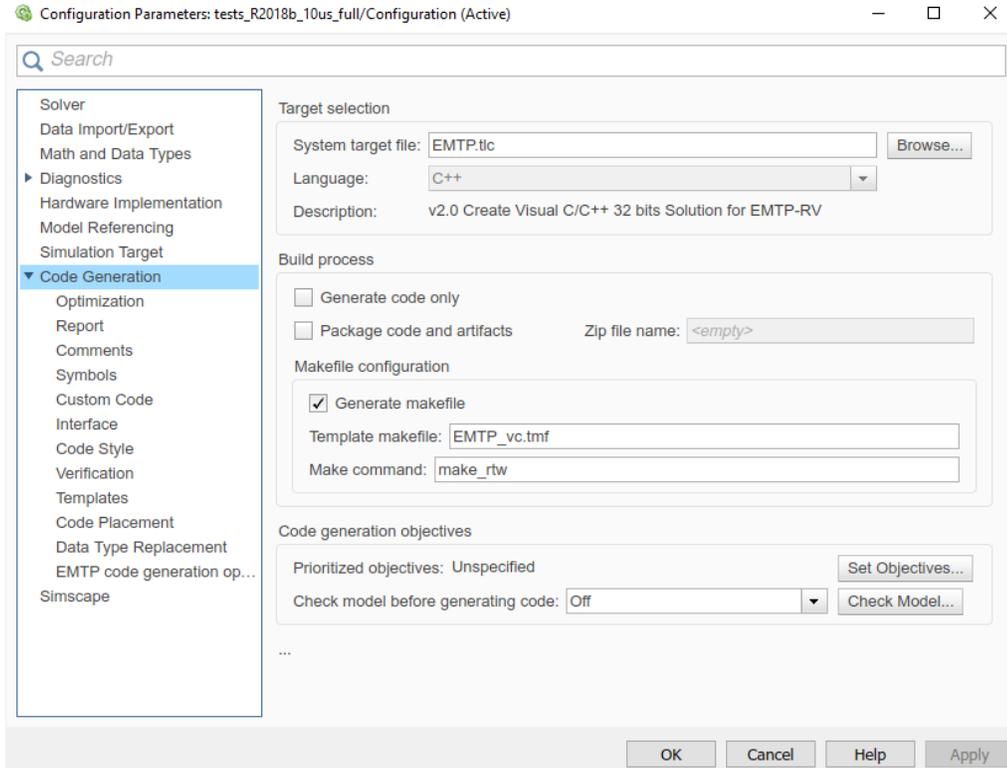


Figure 2 Simulink Model Configuration Parameters, Code Generation.

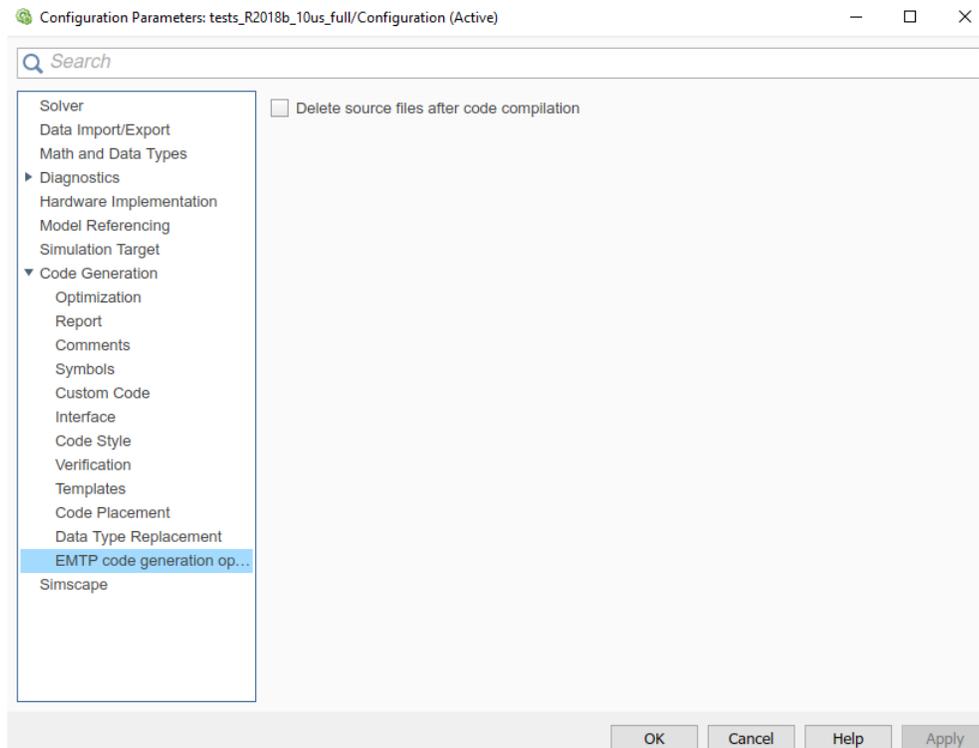


Figure 3 Simulink Model Configuration Parameters, EMTP code generation options.

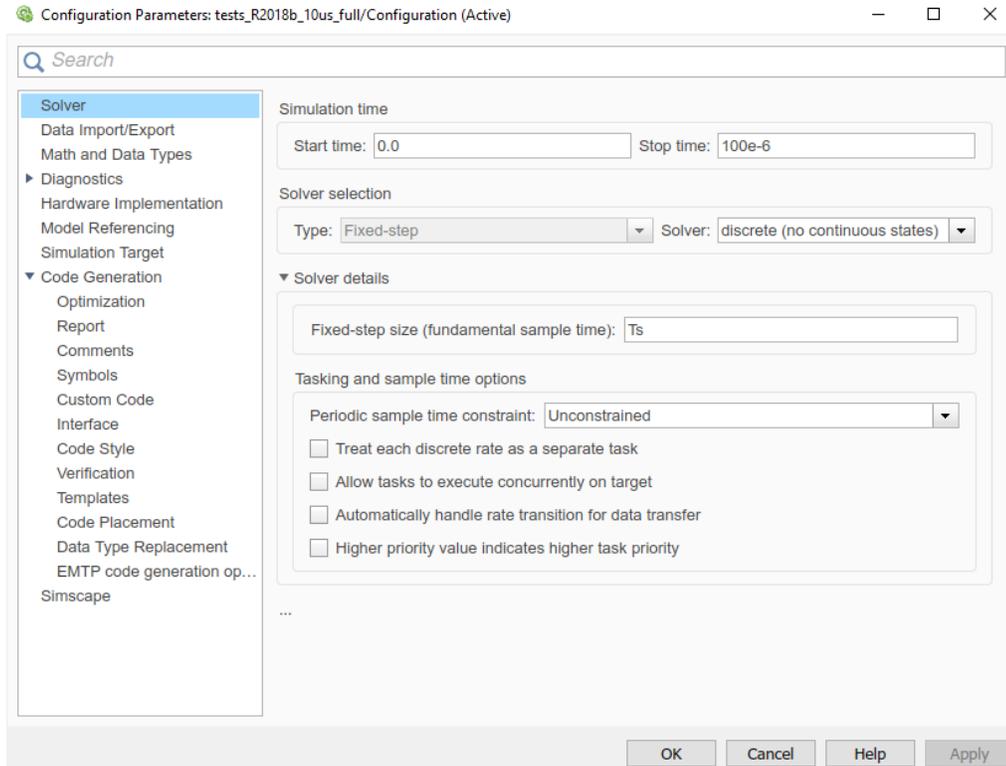


Figure 4 Simulink Model Configuration Parameters, Solver.

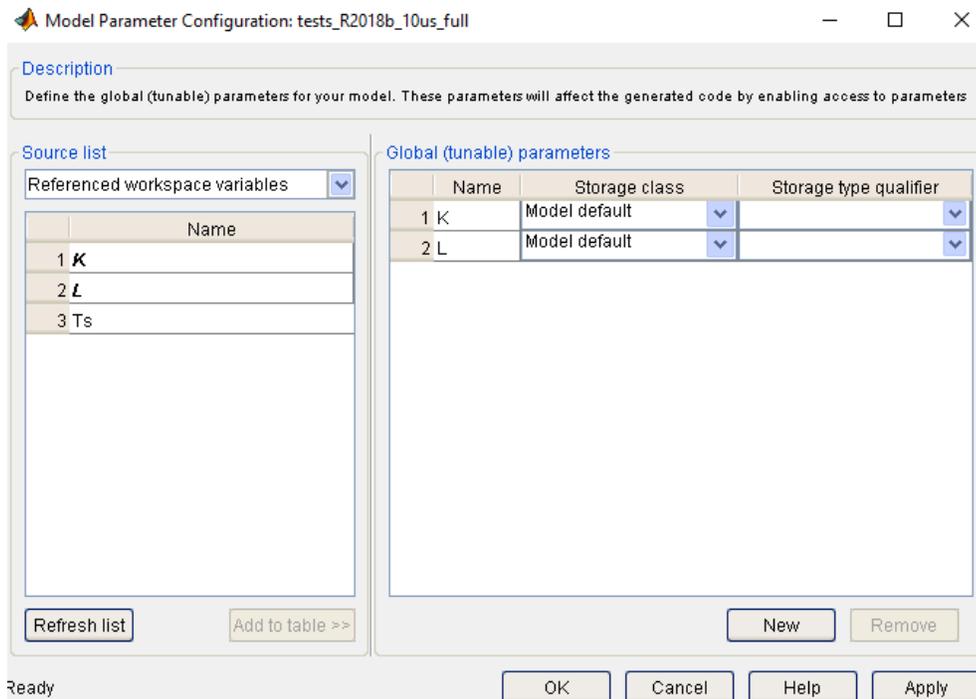


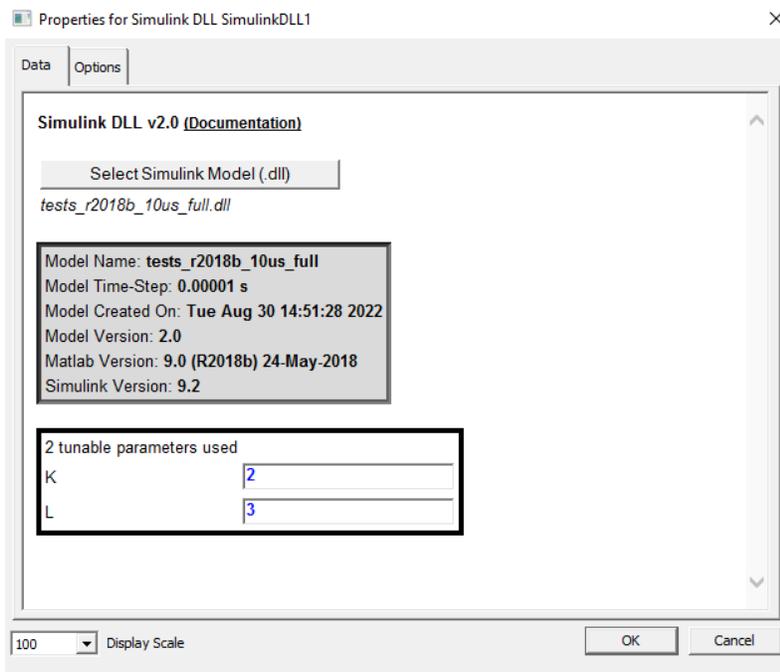
Figure 5 Simulink Model Configuration Parameters, Tunable parameters option for Matlab R2018b or older

3.2.3 Importing the model into EMTWorks

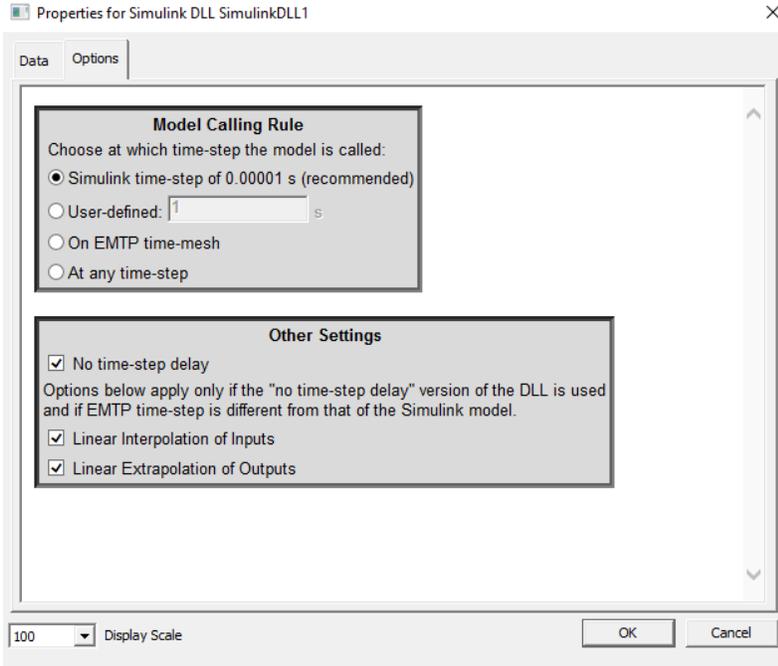
Once the previous DLL code generation steps have been completed, it is possible to import (interface) the DLL into EMTWorks. The user must drag in the device named "Simulink DLL" into the design and double-click on it for selecting the Simulink model. The "Select Simulink Model" button (see Figure 6) is used to browse and select the DLL file ModelName.dll generated in the above steps. In the example of Figure 1 the ModelName.dll is actually tests_r2018b_10us_full.dll.

Once the DLL selection is completed, some model information is displayed (name, time-step, versions of Matlab and Simulink, etc.) and the tunable parameters (if any) are shown (see Figure 6) in the Data tab. The corresponding EMT file EMTPSimulinkDLL.ecf can be found in the folder Examples.

The tunable parameters can become named-values and used in upper level subnetwork masks on EMTWorks side, if needed.



a) Main data tab



b) Options

Figure 6 EMTWorks device “Simulink DLL” data tabs.

When the OK button of Figure 6 is clicked, the EMTWorks interface block (see Figure 7) is automatically drawn with all the appropriate input and output pins and pin types. These are all control (Input or Output) pins.

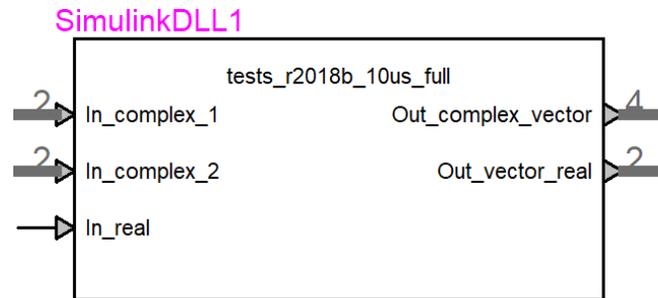


Figure 7 EMTWorks interface block with Simulink DLL.

The interface block of Figure 7 is related to the selected version of the Simulink DLL. If the DLL is recompiled (regenerated) the interface will not be automatically updated with changes in interfacing pins, time-step and tunable parameters on Simulink side.

It is apparent from Figure 7 that vector type ports (pins) on Simulink side become bundle pins on EMTWorks side. The number of pins in each bundle is show on the bundle pin.

It is noticed also that a complex signal on Simulink size is interpreted as a bundle on EMTWorks side. This is illustrated for the Out_complex_vector pin in Figure 8.

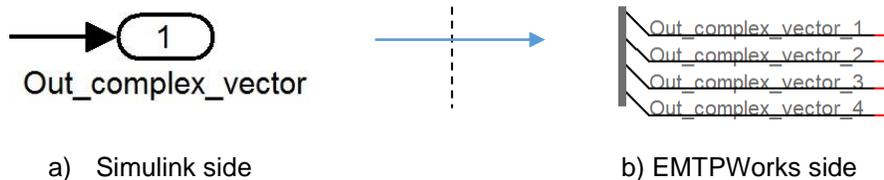


Figure 8 Transformation of a complex Simulink vector (see Figure 1) into EMTWorks bundle.

3.2.4 Rules and limitations

The complexity of Simulink may create various incompatibilities and issues for code generation and interfacing with EMTWorks. Here is a set of known rules that the user must follow.

1. The top level model (design) in Simulink must contain all Input and Output ports for interfacing with EMTWorks. Simulink export is not allowed for a model without input.
2. Tunable parameters
 - a. A tunable parameter is a parameter that can be modified in Simulink during simulation.
 - b. Tunable parameter settings are ignored when referenced models are used in Simulink.
 - c. In some cases the tunable parameters may cause problems by losing tunability. In such cases MATLAB may not necessarily send an error message or the error message may be generic. See the following section for debugging such conditions.
3. S-Function
 - a. There are two types of S-Functions in Simulink: standard S-functions based on C or C++ and inlined S-functions based on Simulink TLC files.
 - b. To be able to use two or more Simulink models (DLLs) on EMTWorks side, it is necessary to use only inlined S-functions. See also MATLAB documentation “Write S-Functions that Support Code Reuse” for other issues, such as non-static variables in S-function codes.
 - c. The multi-instance compatibility problems can be avoided by using two independent DLLs (different names) that can be generated using two different Simulink model file names for the exact same model.
4. Algebraic loops
 - a. Simulink uses an iterative method for solving algebraic loops. A typical algebraic loop is shown in Figure 9.
 - b. It is not possible in Simulink to generate code that can solve algebraic loops. In such cases, the user must artificially break the loops by inserting delays and validate that the block diagram maintains desired functionalities.

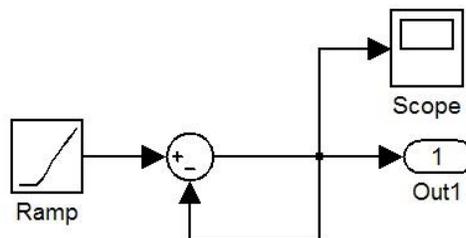


Figure 9 Algebraic loop condition in Simulink

3.2.5 Searching for tunable parameter problems on Simulink side

The safe approach for avoiding tunable parameter problems is to test them on Simulink side before establishing the interface with EMTWorks.

The user can search for all blocks using tunable parameters and test the tunable functionality by changing the parameter to observe its impact on simulation results. The “Inline parameters” option in “Optimization of Signals and Parameters” (see Figure 3) must be unchecked for performing the following tests.

If L is a tunable parameter, it can be located in a model using the CTRL+F keyboard command. The window of Figure 10 with the selected configurations is used in the example of this document. The search results are listed at the bottom of the window. The user can open the dialog box of a listed block, make a change and validate that the parameter is tunable during the simulation by pressing Apply.

The user can search for blocks with masks and validate that tunable parameters are not used in the Initialization tab. A sample case is shown in Figure 12. The variable L (see Figure 12a) is used to define the variable A in a

subsystem mask (see Figure 12b) and the variable A is used for initialization in see Figure 12c. In this case the variable B in see Figure 12c is not tunable, which means that changing L will affect A, but not B! In this example, if the initialization tabs are not using the variable A, the user must continue searching by replacing the searched variable L by the variable A since this variable has inherited tunability.

Further details can be found in MATLAB documentation (see Tunable Expression in Subsystem Mask Dialog Box).

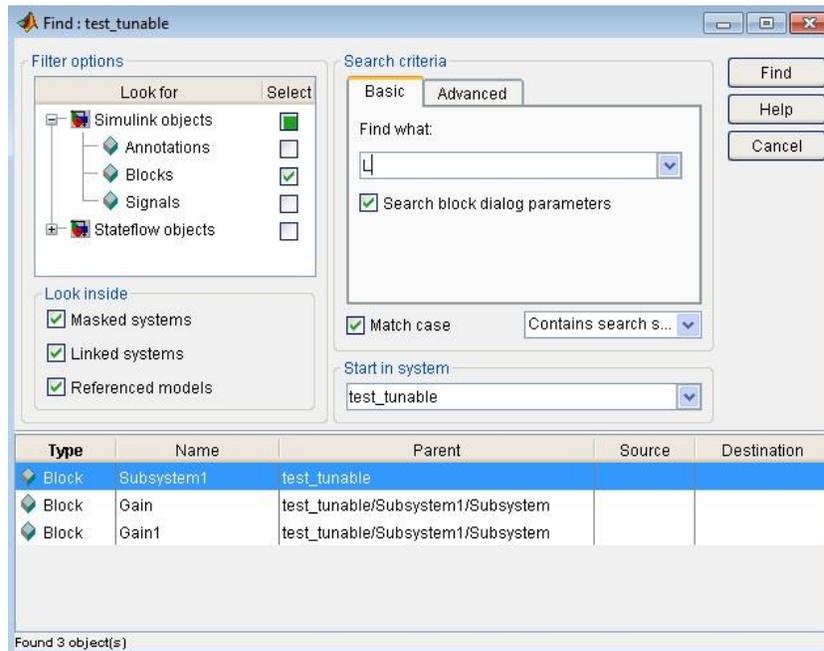


Figure 10 Search window for tunable parameter in Simulink.

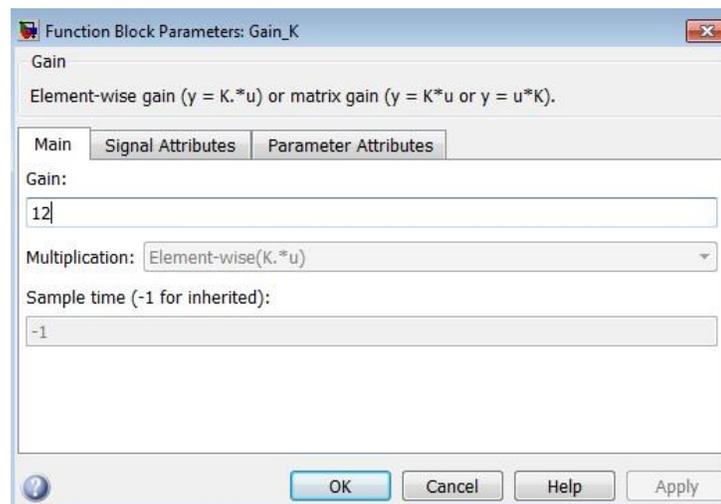
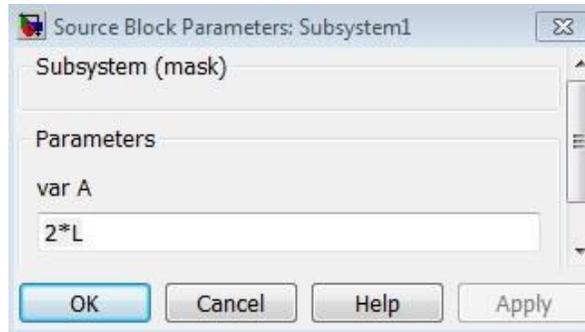
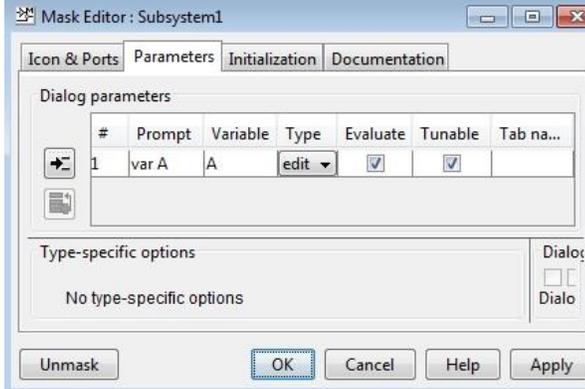


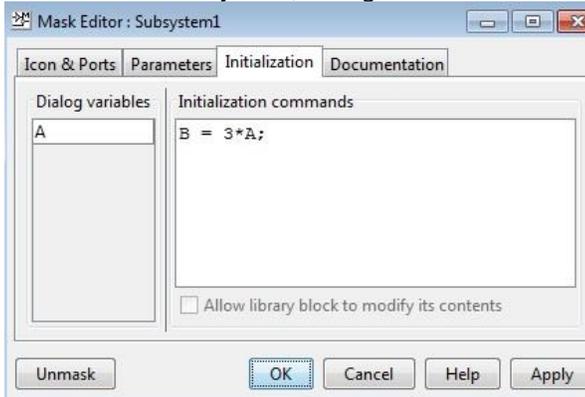
Figure 11 The Gain_K block, the tunable parameter can be replaced (see Gain field) and Applied.



a) Subsystem mask definition with tunable parameter L



b) Mask Editor for Subsystem, dialog Parameters tab



c) Mask Editor for Subsystem, Initialization tab

Figure 12 Simulink Mask with tunable parameter

4 Simulink model options and configurations

See Figure 6 for the Simulink model options

The EMTP numerical integration time-step is named hereinafter Δt_{EMTP} .

The intrinsic time step of the Simulink model, i.e. the one used during the code generation from Simulink is noted $\Delta t_{InternalSimulink}$. This value is not modifiable, it is hard-coded in the DLL.

The one used to call the Simulink model is referred to as $\Delta t_{CallSimulink}$. This value is modifiable in the Model Calling Rule part of the Options tab.

4.1 Model Calling Rule

This option allows an advanced user to choose when EMTP will call the model. Keep in mind that the internal model time step is not modifiable.

The recommended configuration is $\Delta t_{EMTP} = \Delta t_{InternalSimulink} = \Delta t_{CallSimulink}$, but the setup allows to handle different situations and may provide acceptable results in some cases.

Here are possible options for $\Delta t_{CallSimulink}$

4.1.1 Simulink time-step (default option)

In this case, we have $\Delta t_{InternalSimulink} = \Delta t_{CallSimulink}$. If Δt_{EMTP} is set to the same value, we are in the recommended configuration and the Simulink DLL is called at every full EMTP time step. If Δt_{EMTP} is different from Δt_{EMTP} , calls are adapted (see 4.3)

4.1.2 User-defined

Enter a specific value for $\Delta t_{CallSimulink}$.

4.1.3 On EMTP time-mesh

The Simulink DLL is called at every full EMTP time step (Δt_{EMTP}). It does not include calls at intermediate Backward Euler time-points neither calls at all "Simultaneous switching" solutions when re-solving control systems as specified.

4.1.4 At any time-step

The Simulink DLL is called at all solution time-points in EMTP so whenever EMTP requests it.

- a. It includes calls at intermediate Backward Euler time-points in EMTP
 - b. It includes calls at all "Simultaneous switching" solutions when re-solving control systems as specified.
- Such a selection means the Simulink model is time-step independent and can provide accurate response at every call.

4.2 Other Settings

4.2.1 DLL version

By default, the "no time-step delay" version is selected. That means the SDI is called following the solution of EMTP control equations.

If this option is unchecked:

- a) It is expected that simulations results in EMTP will be delayed by the simulation time-step. At startup all Simulink DLL output signals are initialized to zero.

The SDI will be called following the solution of EMTP network equations (as when calling a control diagram in EMTP) and the outputs from an SDI block are used in the network side solution at the following time-point. This means that there is a time-step delay between the network solution on EMTP side and the solution of an SDI block.

- b) In some cases, the performance (in terms of simulation time) is improved, especially if the number of outputs of the model is large

Muti-instance (two or more Simulink models pointing to the same DLL) using different values for this option is not allowed.

4.2.2 Interpolation and Extrapolation options

In case the "no time-step delay" version is selected, it is possible to use the Interpolation and Extrapolation options. These options are enabled by default and relevant only in the case where $\Delta t_{EMTP} \neq \Delta t_{CallSimulink}$. For example, if $\Delta t_{EMTP} = 10\mu s$, $\Delta t_{CallSimulink} = 7\mu s$, and the Interpolation and Extrapolation options are enabled :

- $t = 0\mu s$: the Simulink model is called with inputs values at $t = 0\mu s$
- $t = 10\mu s$: inputs values at $t = 7\mu s$ are estimated by interpolation between inputs values at $t = 0\mu s$ and inputs values at $t = 10\mu s$. Then the Simulink model is called. The values of the outputs obtained are estimated values at $t = 7\mu s$ so extrapolation is done to get estimated outputs values at $t = 0\mu s$.
- Etc.

This must be done with caution and sufficiently good accuracy may not be achieved

4.3 Behaviour when $\Delta t_{EMTP} \neq \Delta t_{CallSimulink}$

Here are some examples:

1. $\Delta t_{CallSimulink} = 10\mu s$ $\Delta t_{EMTP} = 20\mu s$
 - $t = 0\mu s$: the Simulink model is called
 - $t = 20\mu s$: the Simulink model is called twice
 - $t = 40\mu s$: the Simulink model is called twice
 - etc.
2. $\Delta t_{CallSimulink} = 20\mu s$ $\Delta t_{EMTP} = 10\mu s$
 - $t = 0\mu s$: the Simulink model is called
 - $t = 10\mu s$: the Simulink model is not called
 - $t = 20\mu s$: the Simulink model is called
 - $t = 30\mu s$: the Simulink model is not called
 - $t = 40\mu s$: the Simulink model is called
 - etc.

Inputs and outputs values differ depending on whether the Interpolation and Extrapolation options are enabled or not.

5 Compatibility with EMTP simulation options

The “Statistical Options” are compatible with the Simulink DLL.

When the “Simultaneous switching” option is used, it may request to “Re-solve control system equations”. This means that the control system equations and the Simulink DLL are called more than one times at the same simulation time-point. The Simulink DLL is called only once (first solution time-point) except when the option “call at any time-step” is checked (see 4.1)

There is no steady-state solution with a Simulink DLL.