
Getting started with the eDSim electrical simulator

Introduction

eDSim is a fast and powerful electrical simulation tool for SMPS and analog ICs, integrated in the [eDesignSuite](#) tool. It features enhanced accuracy and higher convergence speed for SMPS, enabling a simulation time 10-50x faster than the classical analog SPICE simulators.

The tool is a specific version of the SIMPLIS/SIMetrix simulation environment paired with ST model components, for a full electrical simulation with no limitation of nodes and circuit size as long as you use the models provided by ST.

Design your analog circuit using the [eDesignSuite](#) engine, display a preview of it in few seconds with a full annotated schematic and BOM, and then run the electrical simulation through [eDSim](#) to get fast and accurate simulations and reliable design validation, thus reducing the effort and risks related to hardware prototyping.

Related links

[How to get started using eDSim](#)

[Case study: design and simulate L6983 buck converter operation with eDSim](#)

1 How to start the eDSim simulator

To start the eDSim analog simulator, follow the steps below.

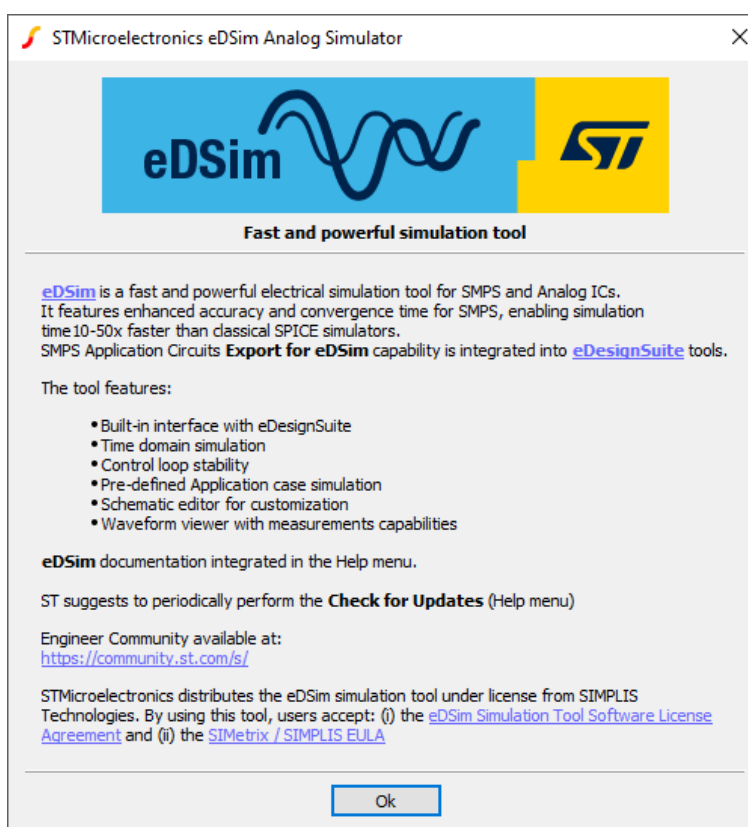
- Step 1.** Double-click on the desktop shortcut after the installation or select eDSim from the start menu of your PC.

Figure 1. eDSim simulator icon



A splash screen then shows the eDSim analog simulator logo and the licensing terms.

Figure 2. eDSim simulator splash screen



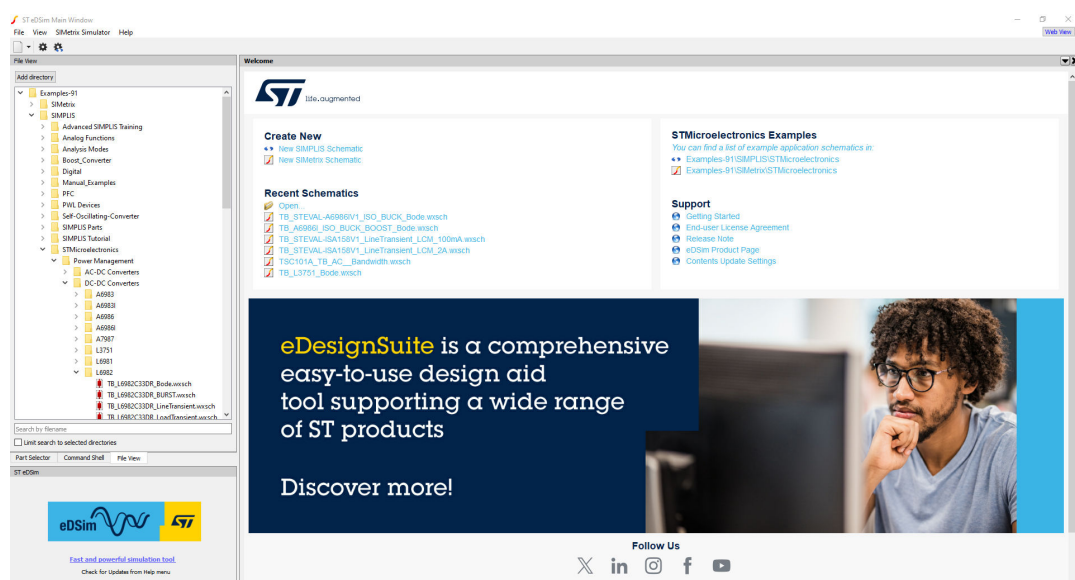
- Step 2.** Click [OK].
The welcome page appears on your screen.

2 Welcome page overview

The welcome page features the following sections:

- **[Create New]** to create a new schematic for the SIMPLIS or SIMetrix environment;
- **[Recent Schematics]** to open all the recent available schematics;
- **[STMicroelectronics Examples]** to obtain a list of example application schematics available in:
 - Examples-91\SIMPLIS\STMicroelectronics
 - Examples-91\SIMetrix\STMicroelectronics
- **[Support]** to get:
 - a **[Getting Started]** (this user manual) that represents a quick introduction to the eDSim tool
 - the **[End-user License Agreement]** (EULA)
 - a technical **[Release Note]** with the last feature enhancements and bug fixes
 - the **[eDSim Product Page]**, which contains the whole eDSim project folder
 - the **[Contents Update Settings]** allows you to check and download the latest available updates

Figure 3. eDSim simulator welcome page



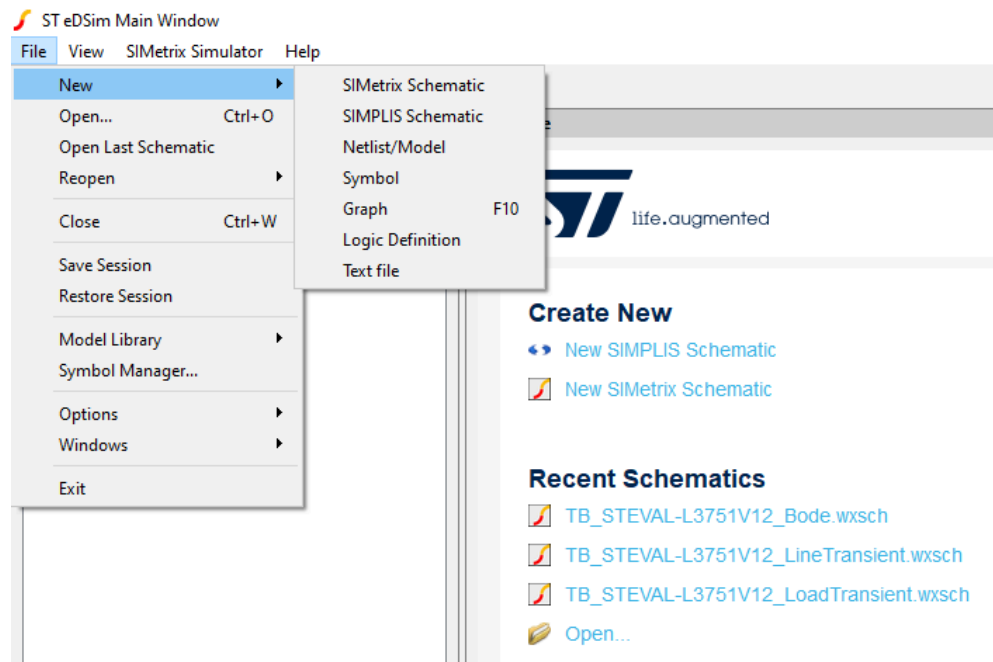
2.1 [Create New]

To create a new schematic or symbol for SIMetrix or SIMPLIS from scratch, follow the procedure below.

Step 1. Choose one of the following options.

- Click on [File]>[New]>[SIMetrix/SIMPLIS Schematic]

Figure 4. eDSim main window - create new



- Use the [Create New] quick link from the welcome page.

Figure 5. eDSim welcome page - create new

Create New

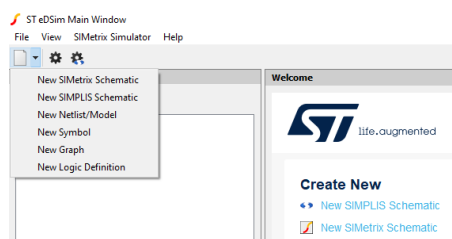
- ↔ New SIMPLIS Schematic
- New SIMetrix Schematic

Recent Schematics

- TB_STEVAL-L3751V12_LoadTransient.wxsch
- Open...

- Click on the [Blank Page] button.

Figure 6. eDSim blank page button - create new



Step 2. Modify Simplis/SIMetrix options parameters by clicking on the [Options/Preferences] and [Simplis Option] buttons next to the [Blank Page] button.

Figure 7. [Blank Page], [Options/Preferences], [Simplis Options] buttons



Figure 8. [Options/Preferences]

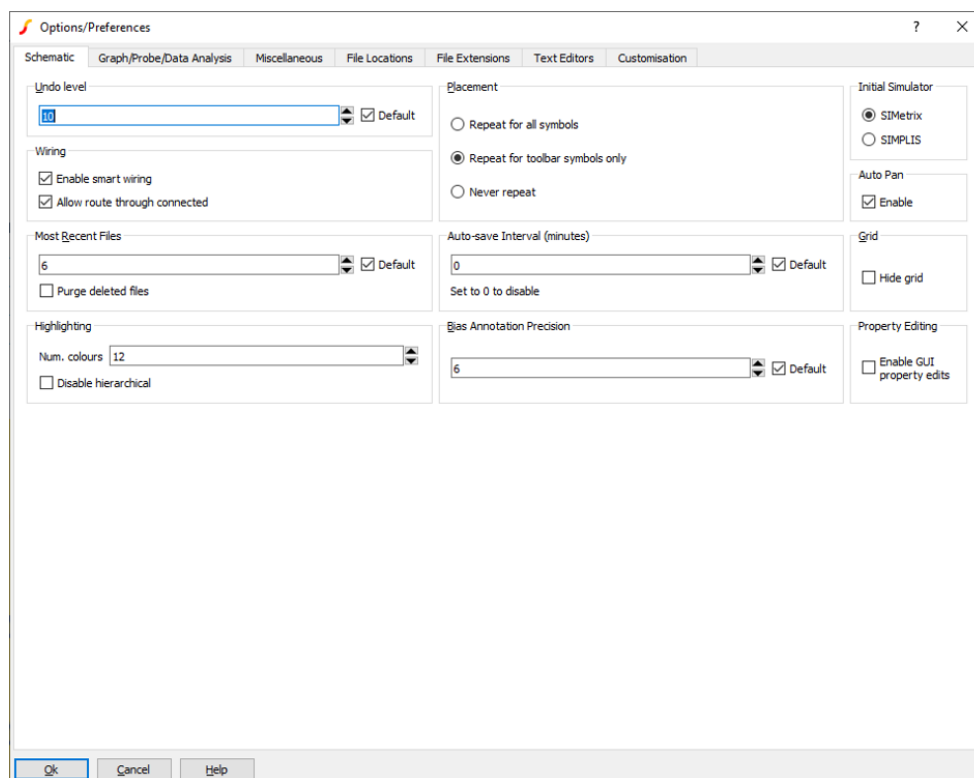
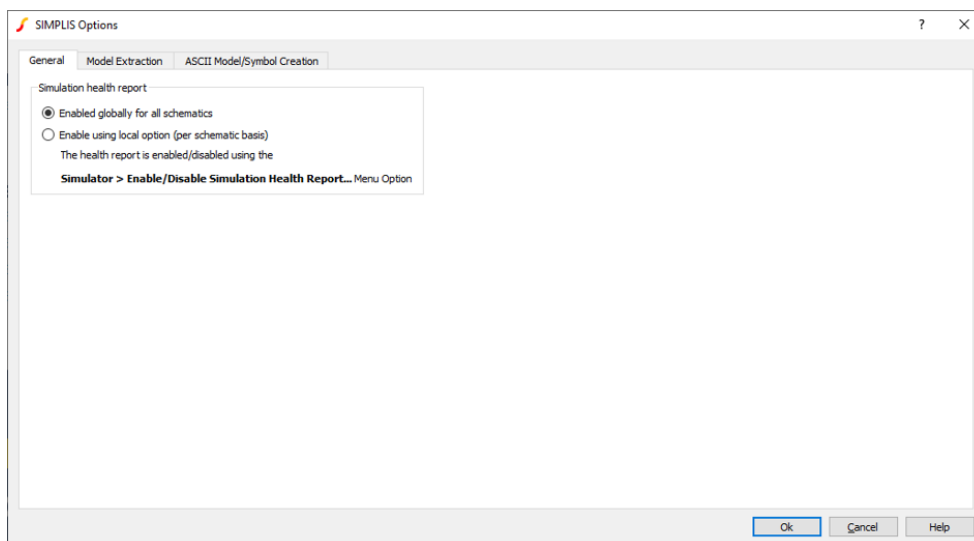


Figure 9. [Simplis Options]



The model list is organized according to the product category. The tool shows only the models compatible with the currently selected simulator (SIMetrix or SIMPLIS).

The screenshot shows the ST-GEM Main Window with the 'Place' menu open. The 'From STMicroelectronics Library' option is selected, and the 'AC-DC Converters' list is displayed. The list includes various components such as A0803, A0804, A0806, A2787, L1751, L6081, L6082, L6083, L6084, L6085, L6086, L6087, L6088, L6089, L6090, L6091, L6092, L6093, L6094, L6095, L6096, L6097, L6098, L6099, L6100, L6101, L6102, L6103, L6104, L6105, L6106, L6107, L6108, L6109, L6110, L6111, L6112, L6113, L6114, L6115, L6116, L6117, L6118, L6119, L6120, L6121, L6122, L6123, L6124, L6125, L6126, L6127, L6128, L6129, L6130, L6131, L6132, L6133, L6134, L6135, L6136, L6137, L6138, L6139, L6140, L6141, L6142, L6143, L6144, L6145, L6146, L6147, L6148, L6149, L6150, L6151, L6152, L6153, L6154, L6155, L6156, L6157, L6158, L6159, L6160, L6161, L6162, L6163, L6164, L6165, L6166, L6167, L6168, L6169, L6170, L6171, L6172, L6173, L6174, L6175, L6176, L6177, L6178, L6179, L6180, L6181, L6182, L6183, L6184, L6185, L6186, L6187, L6188, L6189, L6190, L6191, L6192, L6193, L6194, L6195, L6196, L6197, L6198, L6199, L6200, L6201, L6202, L6203, L6204, L6205, L6206, L6207, L6208, L6209, L6210, L6211, L6212, L6213, L6214, L6215, L6216, L6217, L6218, L6219, L6220, L6221, L6222, L6223, L6224, L6225, L6226, L6227, L6228, L6229, L6230, L6231, L6232, L6233, L6234, L6235, L6236, L6237, L6238, L6239, L6240, L6241, L6242, L6243, L6244, L6245, L6246, L6247, L6248, L6249, L6250, L6251, L6252, L6253, L6254, L6255, L6256, L6257, L6258, L6259, L6260, L6261, L6262, L6263, L6264, L6265, L6266, L6267, L6268, L6269, L6270, L6271, L6272, L6273, L6274, L6275, L6276, L6277, L6278, L6279, L6280, L6281, L6282, L6283, L6284, L6285, L6286, L6287, L6288, L6289, L6290, L6291, L6292, L6293, L6294, L6295, L6296, L6297, L6298, L6299, L6300, L6301, L6302, L6303, L6304, L6305, L6306, L6307, L6308, L6309, L6310, L6311, L6312, L6313, L6314, L6315, L6316, L6317, L6318, L6319, L6320, L6321, L6322, L6323, L6324, L6325, L6326, L6327, L6328, L6329, L6330, L6331, L6332, L6333, L6334, L6335, L6336, L6337, L6338, L6339, L6340, L6341, L6342, L6343, L6344, L6345, L6346, L6347, L6348, L6349, L6350, L6351, L6352, L6353, L6354, L6355, L6356, L6357, L6358, L6359, L6360, L6361, L6362, L6363, L6364, L6365, L6366, L6367, L6368, L6369, L6370, L6371, L6372, L6373, L6374, L6375, L6376, L6377, L6378, L6379, L6380, L6381, L6382, L6383, L6384, L6385, L6386, L6387, L6388, L6389, L6390, L6391, L6392, L6393, L6394, L6395, L6396, L6397, L6398, L6399, L6400, L6401, L6402, L6403, L6404, L6405, L6406, L6407, L6408, L6409, L6410, L6411, L6412, L6413, L6414, L6415, L6416, L6417, L6418, L6419, L6420, L6421, L6422, L6423, L6424, L6425, L6426, L6427, L6428, L6429, L6430, L6431, L6432, L6433, L6434, L6435, L6436, L6437, L6438, L6439, L6440, L6441, L6442, L6443, L6444, L6445, L6446, L6447, L6448, L6449, L6450, L6451, L6452, L6453, L6454, L6455, L6456, L6457, L6458, L6459, L6460, L6461, L6462, L6463, L6464, L6465, L6466, L6467, L6468, L6469, L6470, L6471, L6472, L6473, L6474, L6475, L6476, L6477, L6478, L6479, L6480, L6481, L6482, L6483, L6484, L6485, L6486, L6487, L6488, L6489, L6490, L6491, L6492, L6493, L6494, L6495, L6496, L6497, L6498, L6499, L6500, L6501, L6502, L6503, L6504, L6505, L6506, L6507, L6508, L6509, L6510, L6511, L6512, L6513, L6514, L6515, L6516, L6517, L6518, L6519, L6520, L6521, L6522, L6523, L6524, L6525, L6526, L6527, L6528, L6529, L6530, L6531, L6532, L6533, L6534, L6535, L6536, L6537, L6538, L6539, L6540, L6541, L6542, L6543, L6544, L6545, L6546, L6547, L6548, L6549, L6550, L6551, L6552, L6553, L6554, L6555, L6556, L6557, L6558, L6559, L6560, L6561, L6562, L6563, L6564, L6565, L6566, L6567, L6568, L6569, L6570, L6571, L6572, L6573, L6574, L6575, L6576, L6577, L6578, L6579, L6580, L6581, L6582, L6583, L6584, L6585, L6586, L6587, L6588, L6589, L6590, L6591, L6592, L6593, L6594, L6595, L6596, L6597, L6598, L6599, L6600, L6601, L6602, L6603, L6604, L6605, L6606, L6607, L6608, L6609, L6610, L6611, L6612, L6613, L6614, L6615, L6616, L6617, L6618, L6619, L6620, L6621, L6622, L6623, L6624, L6625, L6626, L6627, L6628, L6629, L6630, L6631, L6632, L6633, L6634, L6635, L6636, L6637, L6638, L6639, L6640, L6641, L6642, L6643, L6644, L6645, L6646, L6647, L6648, L6649, L6650, L6651, L6652, L6653, L6654, L6655, L6656, L6657, L6658, L6659, L6660, L6661, L6662, L6663, L6664, L6665, L6666, L6667, L6668, L6669, L6670, L6671, L6672, L6673, L6674, L6675, L6676, L6677, L6678, L6679, L6680, L6681, L6682, L6683, L6684, L6685, L6686, L6687, L6688, L6689, L6690, L6691, L6692, L6693, L6694, L6695, L6696, L6697, L6698, L6699, L6700, L6701, L6702, L6703, L6704, L6705, L6706, L6707, L6708, L6709, L6710, L6711, L6712, L6713, L6714, L6715, L6716, L6717, L6718, L6719, L6720, L6721, L6722, L6723, L6724, L6725, L6726, L6727, L6728, L6729, L6730, L6731, L6732, L6733, L6734, L6735, L6736, L6737, L6738, L6739, L6740, L6741, L6742, L6743, L6744, L6

The **[Select device]** window allows you to place a device.

Select Device

* Recently Added Models *

* All User Models *

* All Models *

Capacitors

Diode

Discrete Time Filters

Ferrite Beads

Gates

IGBTs

Inductors

N-channel JFET

NMOS

NPN

P-channel JFET

PMOS

PNP

PSU Controllers

Schottky Diodes

Transim Load

Transim Source

Voltage References

Zener Diodes

L6901C3DR

L6901C3DR_for_Bode

L6901C50DR

L6901C50DR_for_Bode

L6901CDR

L6901CDR_for_Bode

L6901N33DR

L6901N33DR_for_Bode

L6901N50DR

L6901N50DR_for_Bode

L6901NDR

L6901NDR_for_Bode

L6902C33DR

L6902C33DR_for_Bode

L6902C50DR

L6902C50DR_for_Bode

L6902CDR

L6902CDR_for_Bode

L6902N33DR

L6902N33DR_for_Bode

L6902N50DR

L6902N50DR_for_Bode

L6902NDR

L6902NDR_for_Bode

L6903C3QTR

L6903C3QTR_for_Bode

L6903C50QTR

L6903C50QTR_for_Bode

L6903CQTR

L6903CQTR_for_Bode

L6903N3QTR

L6903N3QTR_for_Bode

L6903N5QTR

L6903N5QTR_for_Bode

L6903NQTR

L6903NQTR_for_Bode

L7983PU33R

L7983PU33R_for_Bode

L7983PU50R

L7983PU50R_for_Bode

L7983PUR

L7983PUR_for_Bode

STPS01AIR

STPS01AIR_for_Bode

STPS01BIR

STPS01BIR_for_Bode

STPS01CIR

STPS01CIR_for_Bode

STPS01DIR

STPS01DIR_for_Bode

STPS01EIR

STPS01EIR_for_Bode

STPS01FIR

STPS01FIR_for_Bode

STPS01GIR

STPS01GIR_for_Bode

STPS02ACQTR

STPS02BQTR

STPS02BQTR_for_Bode

STPS02CQTR

STPS02CQTR_for_Bode

STPS02DQTR

STPS02DQTR_for_Bode

STPS03A1QTR

STPS03A1QTR_for_Bode

STPS03A3QTR

STPS03A3QTR_for_Bode

UC1825

UC1846

UC3842

UC3843

UC3844

UC3845

Filter

Back Cancel Help

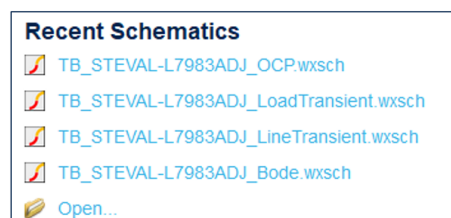
C:\Program Files\Altium\Altium Designer\support\models\psu\psu_controllers.lib

On the top menu bar, you can find a list of available devices, such as capacitors, resistors, inductors, which you can use to design your project.
You can also find a set of probes that allow you to display all the simulation waveforms.

2.2 [Recent Schematics]

This section shows you all the recent opened schematics.

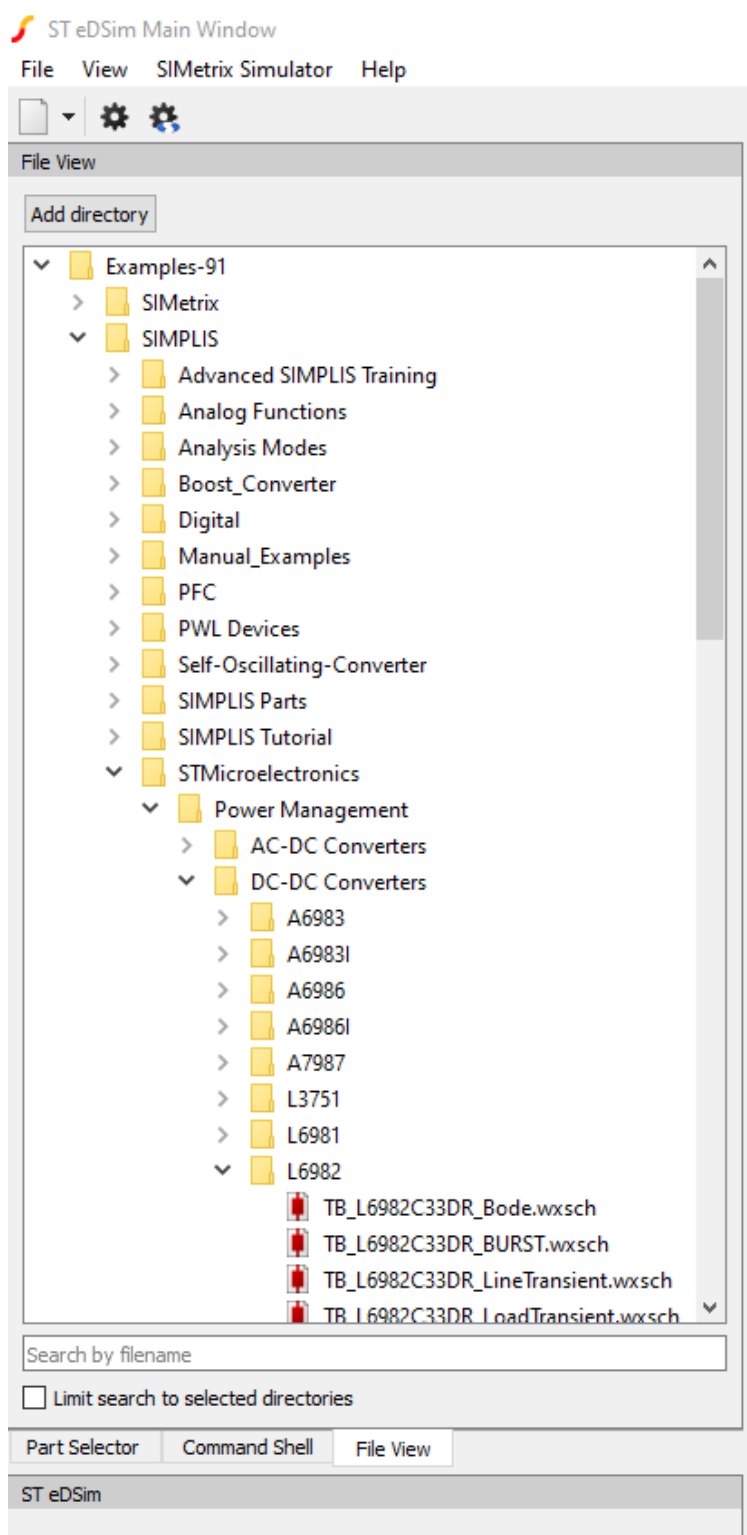
Figure 13. [Recent Schematics] section



3 Predefined ST application test benches

eDSim provides you a list of ST application schematics that you can partially modify according to your needs: access them from the **[File View]** tab in the STMicroelectronics folder under the SIMetrix or SIMPLIS folders, as shown in the figure below taking into account, for example, SIMPLIS.

Figure 14. eDSim directory and folders

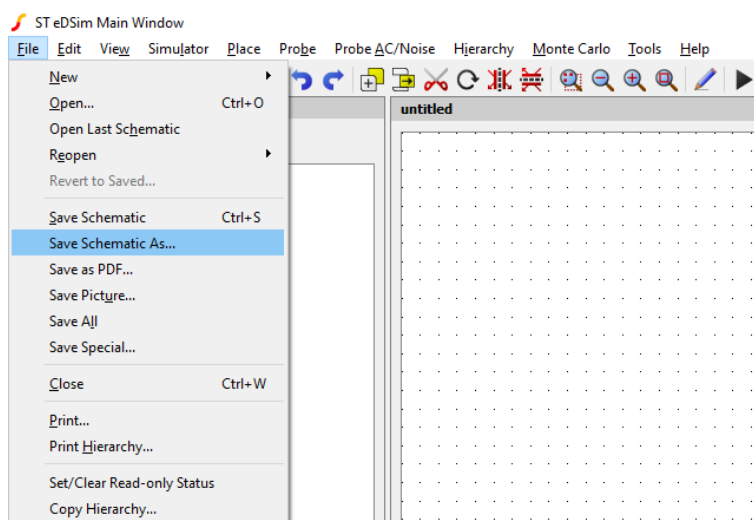


All these schematics are available in your local [Documents] folder, in [ST eDSim]>[Examples].

Important: *If you modify any of these schematics, we highly recommend you to save a copy of the file [File]>[Save Schematics As] to avoid overwriting the default version.*

If you want to save a newly created schematic, select [File]>[Save Schematic As].

Figure 15. Saving a local copy of the schematic

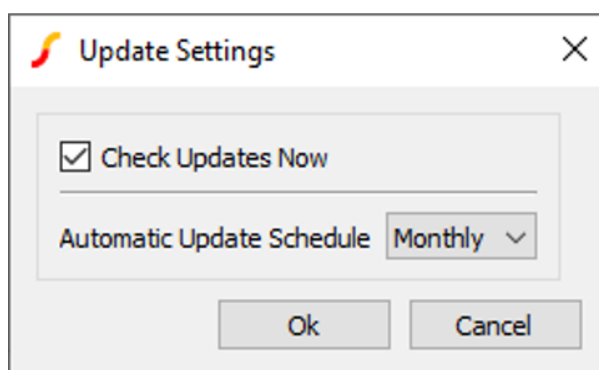


4 Setting the update schedule

The ST schematic library is subject to continuous updates. Thus, you need to set up the eDSim analog simulator to check for any available updates, to get the latest patches, models, and example application schematics.

Step 1. Select **[Help]** on the eDSim command shell window and tick the **[Check Updates Now]** box.

Figure 16. [Update Settings] window



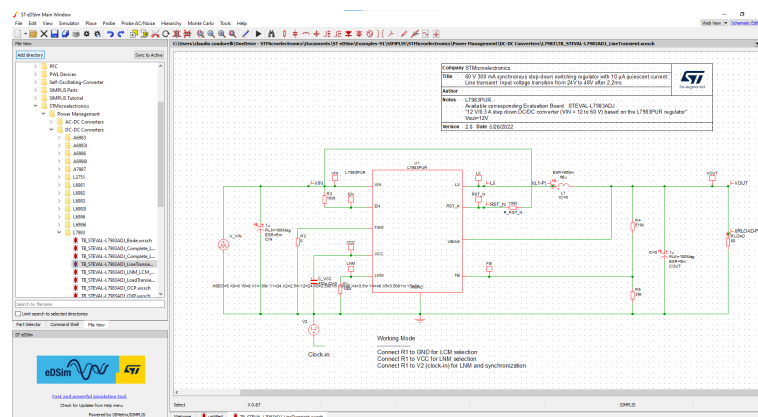
Step 2. To set the desired update schedule, click **[Help][Check Updates Now]** and change the **[Automatic Update Schedule]** to **[Daily]**, **[Weekly]**, or **[Monthly]**.

5 Running the simulation

To start the simulation phase and run the default analysis, follow the procedure below.

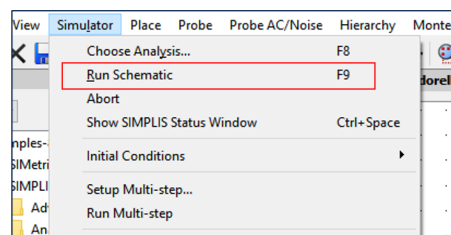
Step 1. Open and customize an ST schematic.

Figure 17. eDSim - starting the simulation



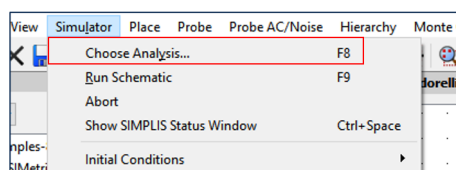
Step 2. Press the F9 key or select [Simulator]>[Run Schematic] to run the schematic simulation.

Figure 18. eDSim - running the simulation



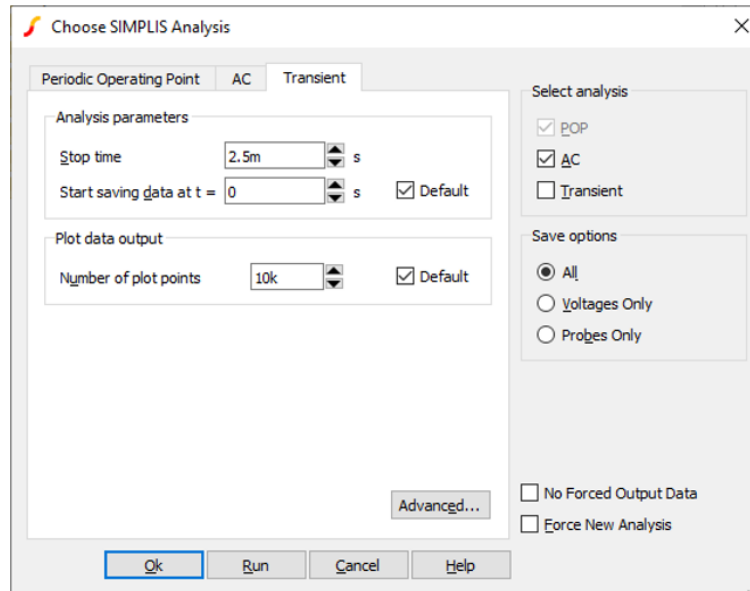
Step 3. Press the F8 key to open the [Choose Analysis] window and modify the analysis type to perform.

Figure 19. eDSim - [Choose Analysis] window



Step 4. Set up the desired parameters.

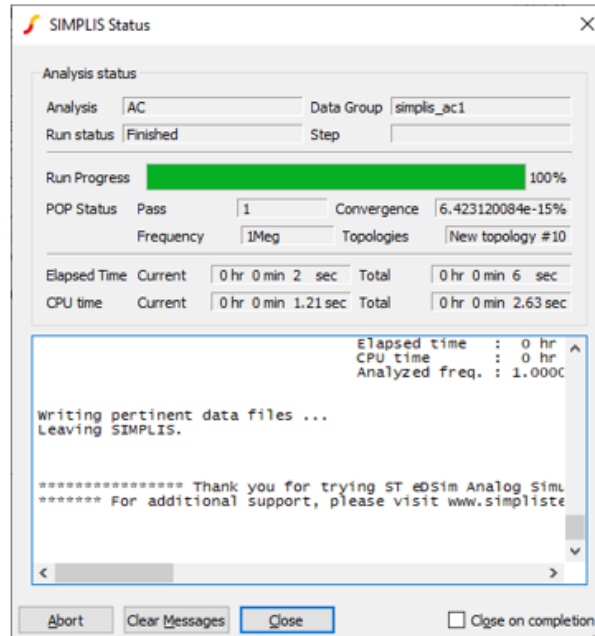
Figure 20. eDSim - [Choose Analysis] settings



5.1 Simulation results

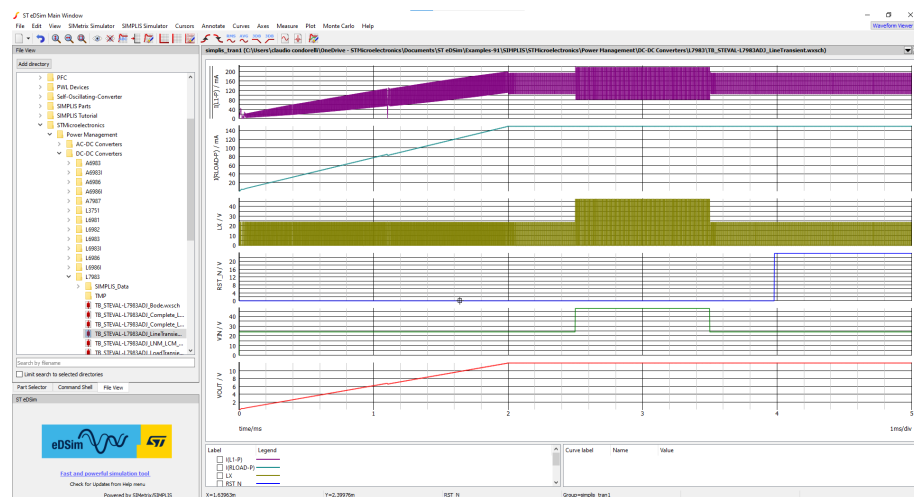
Once you have launched a new simulation, a simulation status window automatically appears. It shows the evolution process of your simulation.

Figure 21. eDSim - simulation status



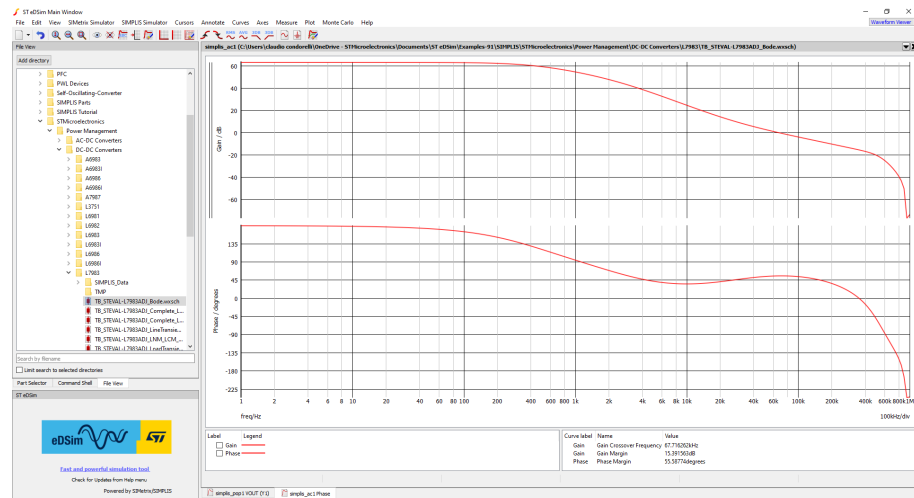
The eDSim analog simulator performs the time domain analysis, which allows the designers to evaluate voltage limits, threshold, and overshoot values.

Figure 22. eDSim - time domain simulation results



eDSim also allows simulating the frequency domain for a proper small signal analysis.

Figure 23. eDSim - Bode simulation results

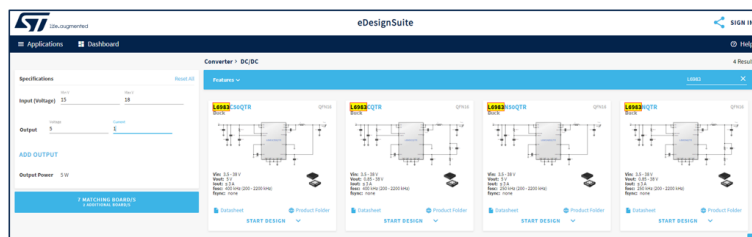


6 eDSim interaction with eDesignSuite

eDesignSuite is an easy-to-use comprehensive online software suite. It helps the customers define their needs and transform their application requirements into satisfactory solutions based on the wide range of ST products. There is a strict connection between the eDSim analog simulator tool and eDesignSuite. Once you have selected the solution that perfectly fits your design specification, you can get all the simulation details using the eDSim simulator.

- Step 1.** Select your design by adding the I/O voltages and current values.
The tool suggests the solutions that match your requirement.

Figure 24. eDesignSuite interface



Once selected on one of the proposed applications, you have the opportunity.

- Step 2.** Run eDSim to analyze, modify, and get more details on the project.
Step 3. Click on [Export for eDSim].

Figure 25. eDesignSuite - IC interface



A popup window appears to help you to install the eDSim tool on your PC and to download the *.zip file of the project started on eDesignSuite.

Figure 26. eDesignSuite - popup window



Step 4. Unzip the file and follow the instructions of the readme.txt.

Figure 27. eDesignSuite - zip folder content

Name	Date modified	Type	Size
 eDSim.bat	20-Apr-21 11:20	Windows Batch File	1 KB
 L6983NQTR.compvalues.log	20-Apr-21 13:33	Text Document	2 KB
 L6983NQTR.compvalues.txt	20-Apr-21 11:20	Text Document	1 KB
 L6983NQTR.sxsch	20-Apr-21 11:20	SIMetrix Schematic	101 KB
 L6983NQTR.sxsch.lck	20-Apr-21 13:33	LCK File	1 KB
 readme.txt	20-Apr-21 11:20	Text Document	1 KB

Step 5. Clicking on the eDSim.bat file.
The eDSim tool automatically loads the solution that you have chosen and modified on eDesignSuite.

7 eDSim limitations

eDSim is a customized edition of the SIMPLIS/SIMetrix simulation environment paired with the ST model specific components. It allows a full electrical simulation with no limitation to the node count and circuit size as long as you use the ST provided models. If you create a new schematic or customize any of the ST available projects, the number of modifications is limited, as detailed in the following sections.

7.1 SIMPLIS circuit size limits

1. A maximum of 15 state variables. Each capacitor or inductor requires one state variable. Each time-varying or small-signal AC source requires one state variable, except for sinusoidal or cosinusoidal sources, which require two state variables per source
2. A maximum of 10 capacitors or inductors combined
3. A maximum of 6 switches (simple or transistor)
4. A maximum of 6 logic gates
5. A maximum of 26 states. Each PWL element, switch, time-varying source, and logic gate requires one state
6. A maximum of 100 new topologies, which are enough for simple switching circuits that use simple models only. More complex circuits or circuits that have more complicated models may exceed this limit

7.2 SIMetrix circuit size limits

For the SIMetrix simulator, the simulated circuit size limits are:

- 140 analog nodes (internal and external)
- 360 digital nodes
- 720 digital ports
- 300 digital components
- 360 digital outputs

Revision history

Table 1. Document revision history

Date	Revision	Changes
19-Apr-2022	1	Initial release.
04-Jul-2022	2	Added links to related videos in the introduction.
16-Dec-2022	3	Updated Section 1 How to start the eDSim simulator, Section 2 Welcome page overview, Section 2.1 [Create New], Section 3 Predefined ST application test benches, Section 5 Running the simulation and Section 5.1 Simulation results.
12-Jul-2023	4	Updated Section 1 How to start the eDSim simulator and Section 2 Welcome page overview.
23-Nov-2023	5	Updated Section 1 How to start the eDSim simulator, Section 2 Welcome page overview, Section 2.1 [Create New] and Section 3 Predefined ST application test benches.
08-May-2024	6	Updated Section 2 Welcome page overview, Section 2.1 [Create New], Section 5 Running the simulation and Section 5.1 Simulation results.

Contents

1	How to start the eDSim simulator	2
2	Welcome page overview	3
2.1	[Create New]	4
2.2	[Recent Schematics]	7
3	Predefined ST application test benches	8
4	Setting the update schedule	10
5	Running the simulation	11
5.1	Simulation results	13
6	eDSim interaction with eDesignSuite	15
7	eDSim limitations	17
7.1	SIMPLIS circuit size limits	17
7.2	SIMetrix circuit size limits	17
	Revision history	18
	List of tables	20
	List of figures	21

List of tables

Table 1.	Document revision history	18
----------	-------------------------------------	----

List of figures

Figure 1.	eDSim simulator icon.	2
Figure 2.	eDSim simulator splash screen.	2
Figure 3.	eDSim simulator welcome page	3
Figure 4.	eDSim main window - create new	4
Figure 5.	eDSim welcome page - create new	4
Figure 6.	eDSim blank page button - create new	4
Figure 7.	[Blank Page], [Options/Preferences], [Simplis Options] buttons	5
Figure 8.	[Options/Preferences]	5
Figure 9.	[Simplis Options].	5
Figure 10.	[Place]>[From STMicroelectronics Library].	6
Figure 11.	Select and place a device	6
Figure 12.	Place buttons	6
Figure 13.	[Recent Schematics] section.	7
Figure 14.	eDSim directory and folders	8
Figure 15.	Saving a local copy of the schematic.	9
Figure 16.	[Update Settings] window	10
Figure 17.	eDSim - starting the simulation	11
Figure 18.	eDSim - running the simulation	11
Figure 19.	eDSim - [Choose Analysis] window	11
Figure 20.	eDSim - [Choose Analysis] settings.	12
Figure 21.	eDSim - simulation status.	13
Figure 22.	eDSim - time domain simulation results	13
Figure 23.	eDSim - Bode simulation results	14
Figure 24.	eDesignSuite interface.	15
Figure 25.	eDesignSuite - IC interface.	15
Figure 26.	eDesignSuite - popup window	15
Figure 27.	eDesignSuite - zip folder content.	16

IMPORTANT NOTICE – READ CAREFULLY

STMicroelectronics NV and its subsidiaries ("ST") reserve the right to make changes, corrections, enhancements, modifications, and improvements to ST products and/or to this document at any time without notice. Purchasers should obtain the latest relevant information on ST products before placing orders. ST products are sold pursuant to ST's terms and conditions of sale in place at the time of order acknowledgment.

Purchasers are solely responsible for the choice, selection, and use of ST products and ST assumes no liability for application assistance or the design of purchasers' products.

No license, express or implied, to any intellectual property right is granted by ST herein.

Resale of ST products with provisions different from the information set forth herein shall void any warranty granted by ST for such product.

ST and the ST logo are trademarks of ST. For additional information about ST trademarks, refer to www.st.com/trademarks. All other product or service names are the property of their respective owners.

Information in this document supersedes and replaces information previously supplied in any prior versions of this document.

© 2024 STMicroelectronics – All rights reserved