g.tec medical engineering GmbH
4521 Schiedlberg, Sierningstrasse 14, Austria
Tel.: (43)-7251-22240-0
Fax: (43)-7251-22240-39
office@gtec.at, http://www.gtec.at

Simulink Highspeed On-line Processing

USER MANUAL V3.15.01

CONTENT:

TO THE READER

Welcome to the medical and electrical engineering world of g.tec!
Discover the only professional biomedical signal processing platform under MATLAB and Simulink. Your ingenuity finds the appropriate tools in the g.tec elements and systems. Choose and combine flexibly the elements for biosignal amplification, signal processing and stimulation to perform even real-time feedback.

Our team is prepared to find the better solution for your needs.

Take advantage of our experience!

Dr. Christoph Guger   Dr. Guenter Edlinger

Researcher and Developer

Reduce development time for sophisticated real-time applications from month to hours.
Integrate g.tec's open platform seamlessly into your processing system.
g.tec's rapid prototyping environment encourages your creativity.

Scientist

Open new research fields with amazing feedback experiments.
Process your EEG/ECG/EMG/EOG data with g.tec's biosignal analyzing tools.
Concentrate on your core problems when relying on g.tec's new software features like ICA, AAR or online Hjorth's source derivation.

Study design and data analysis

You are planning an experimental study in the field of brain or life sciences? We can offer consultation in experimental planning, hardware and software selection and can even do the measurements for you. If you have already collected EEG/ECG/EMG/EOG, g.tec can analyze the data starting from artifact control, do feature extraction and prepare the results ready for publication.

Related Products

g.tec provides several biosignal analysis elements that are especially relevant to the kinds of tasks you perform with g.tec Highspeed.

For more detailed information on any of our elements, up-dates or new extensions please visit our homepage www.gtec.at or just send us an email to office@gtec.at

Conventions

Item	Format	Example
MATLAB code	`Courier`	to start Simulink, type `simulink`
String variables	*Courier italics*	`set(P_C,'`*`PropertyName`*`',...)`
Menu items	**Boldface**	Select **Save** from the **File** menu

Installation and Configuration

This chapter includes the following sections:

Hardware and Software Requirements

Hardware Requirements

g.tec Highspeed requires a PC compatible desktop, notebook workstation running Microsoft Windows.

The table below lists optimal settings:

Hardware	**Properties**
CPU	Pentium working at 3000 MHz
Harddisk	100 gigabyte
RAM	8 gigabyte
USB 2.0 high speed port	One free USB port for g.HIamp (EHCI) and one for the Hardlock Dongle

Software Requirements

g.tec Highspeed requires the installation of the g.HIamp driver which is shipped with the device, MATLAB, Simulink and Signal Processing Blockset. Make sure that the MATLAB installation works correctly before installing the g.tec Highspeed software. Depending on your Windows operating system administrator rights might be necessary for the installation.

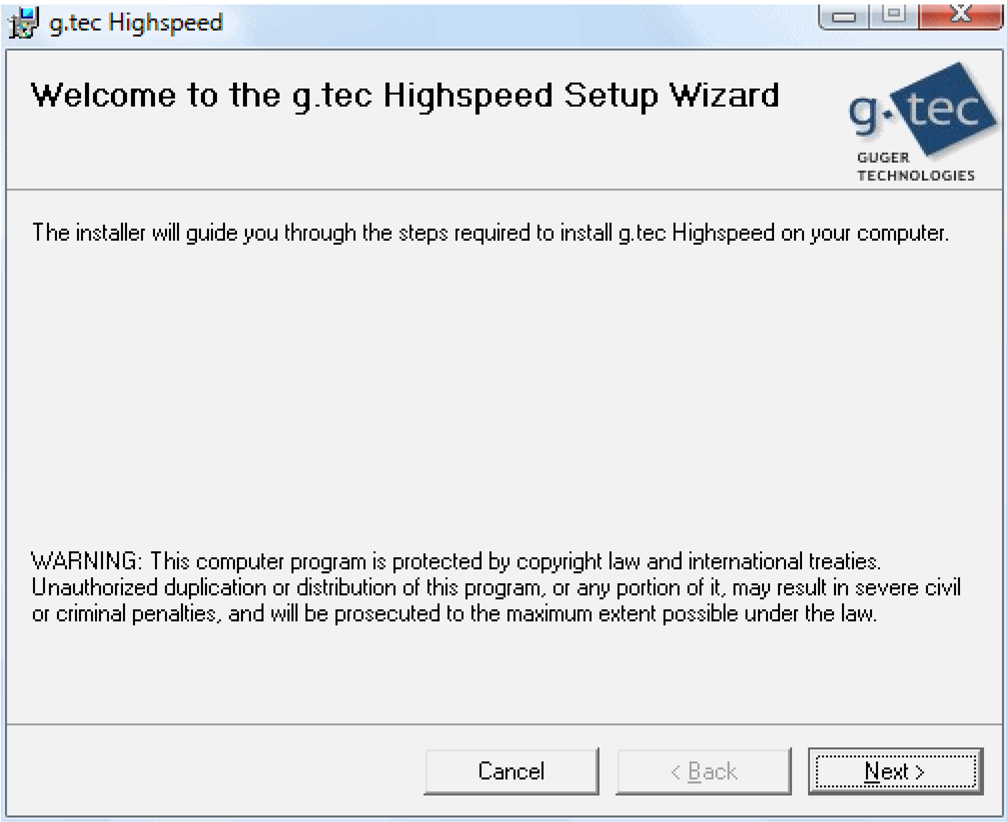
Software	**Version**
g.HIamp Driver	2.14.02 (shipped with g.HIamp)
MATLAB	Release 2014a
Simulink	Release 2014a
Signal Processing Blockset	Release 2014a
Windows	Windows 7 Professional Service Pack 1 English Win64
Acrobat Reader	11.0.04
.net Common Language Runtime	4

Installation from a CD

The installation consists of three steps:

1. Installation of g.tec Highspeed

   Insert the g.tec product CD into the CD-drive and change to the `g.HIsys\g.HIsys Highspeed Online processing for SIMULINK` directory of your CD-drive and double-click on the `Setup.exe` file. The installation starts and displays the welcome message. Follow the instructions on the screen.

   Please read the License Agreement for g.tec Highspeed and if you agree with the terms, click **I Agree** and **Next**.

2. Installation of the Hardlock

   The driver software for the Hardlock is installed automatically via Windows update, when the computer is connected to the internet. For manual installation click the `HASPUserSetup.exe` in the Prerequisites/HaspHL folder on your g.tec CD.

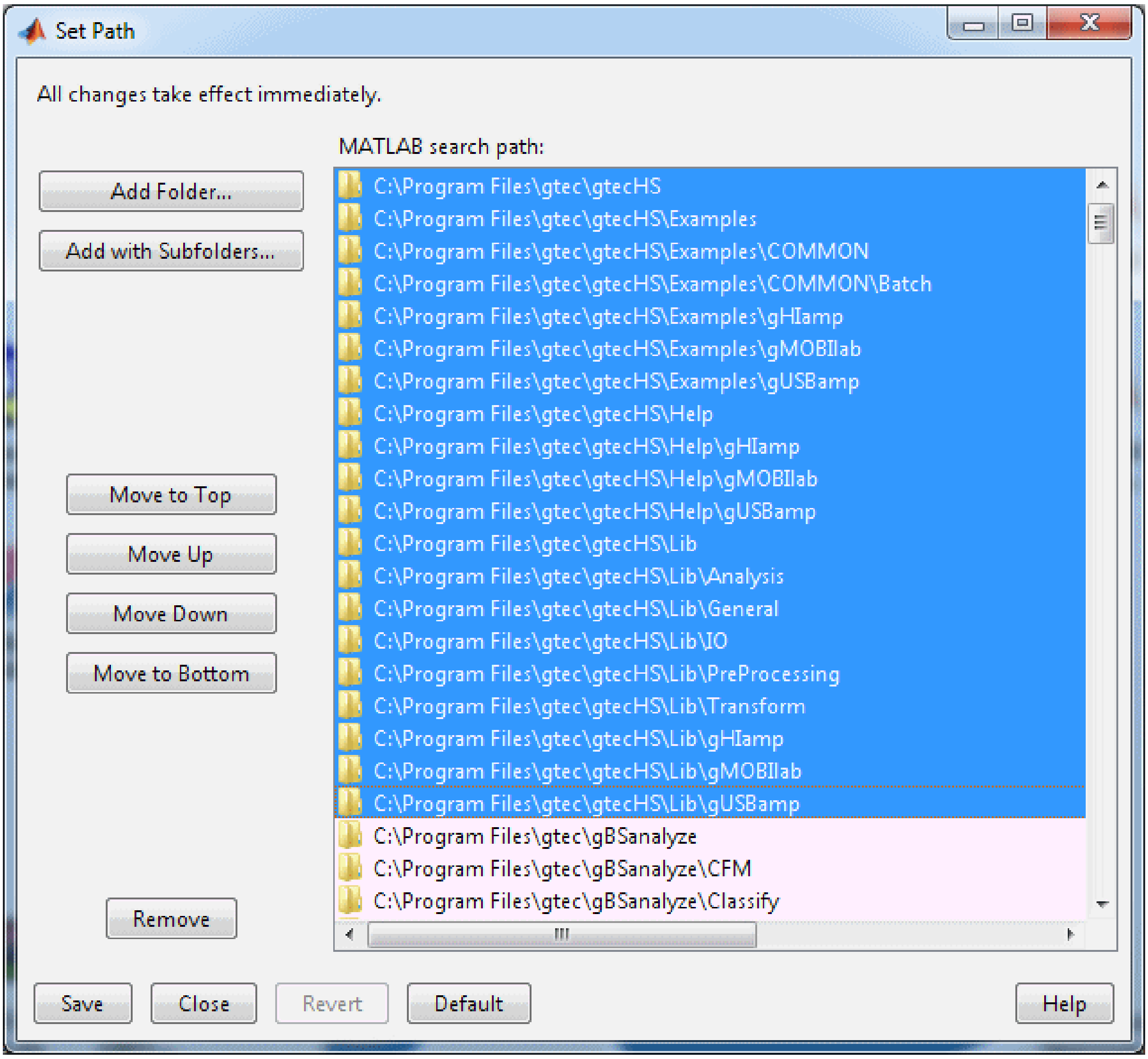
   Note that you have to uninstall the Hardlock driver separately if you uninstall the g.HIamp Highspeed On-line Processing software.

3. Set MATLAB path

To make the path settings start MATLAB and open the **Set Path** window from the **File** menu. Then click on the **Add with Subfolders** button and select

```
C:\Program Files\gtec\gtecHS
```

to add all subdirectories:



Click **Save** and **Close** to finish the installation.

Insert the Hardlock into a free USB slot of your PC or notebook. The light must be on if the installation was successful.

Files on your Computer

g.HIamp Highspeed files - are stored under (it is assumed that the default path setting is used)

`C:\Program Files\gtec\gtecHS\Lib\gHIamp`

Example models - are stored in the subdirectory

`C:\Program Files\gtec\gtecHS\Examples\gHIamp`

Every example model is preconfigured with a g.HIamp configuration stored with the model. Starting the model without configuring g.HIamp applies this configuration. Changing the configuration and saving the model afterwards will change the stored configuration.

Help files - are stored under

`C:\Program Files\gtec\gtecHS\Help\gHIamp`

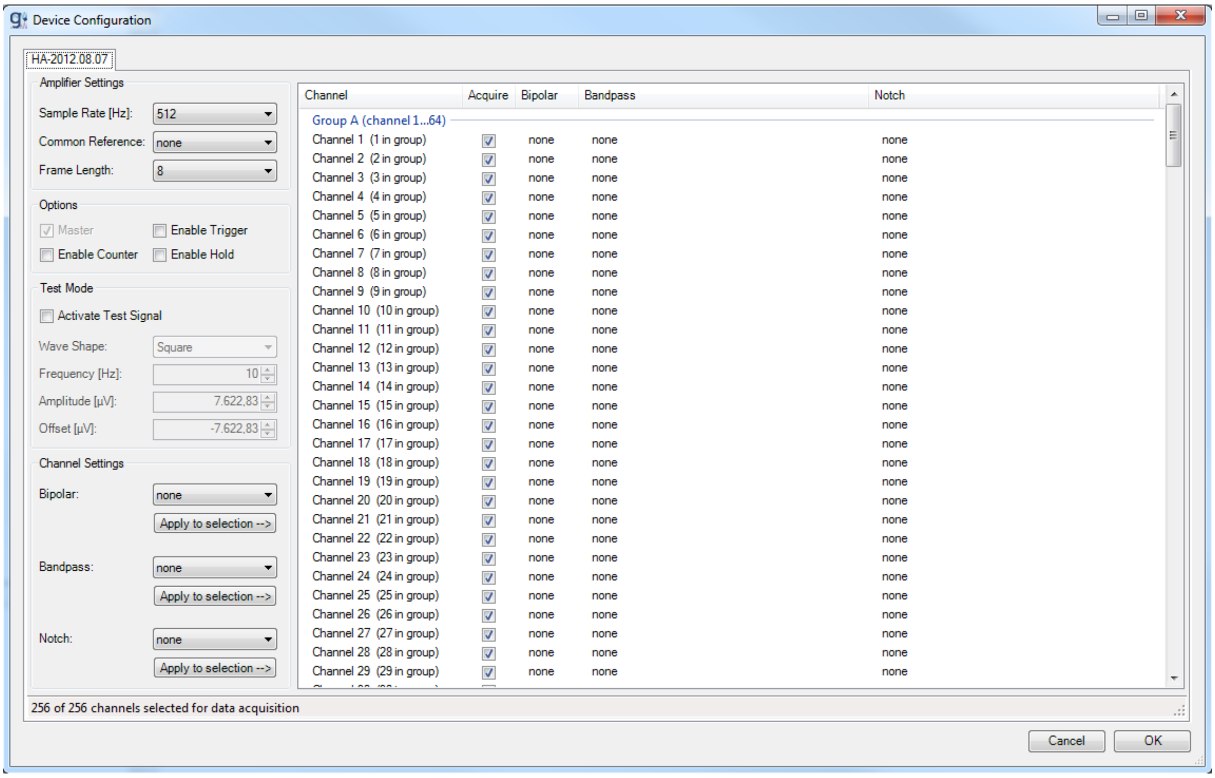
g.HIamp Highspeed Block

The g.HIamp Highspeed block provides a graphical interface to the g.HIamp hardware which can be used under Simulink to specify the amplifier properties and to acquire the data.

Description

The g.HIamp block output signal provides the biosignal data. The data format is single (float32) and it is scaled in µV. If all analog input channels (256) and the trigger signal (1) are acquired the line width is 257. Use a **Demux** block to de-multiplex the channels.

Dialog Box



Specify Amplifier Settings

Sample Rate [Hz]	specify the sampling frequency of the g.HIamp in Hz
Common Reference	set the common reference for all channels
Frame Length	specify the buffering block size

Options

Enable Counter	show a counter on first recorded channel which is incremented with every block transmitted to the PC. Overruns at 1000000.
Enable Trigger	scan the digital trigger channel with the analog inputs
Master	set the amplifier to master mode if multiple units are used
Enable Hold	enable signal hold

Test Mode

Activate Test Signal	apply internal test signal to all inputs
Analog output	generate a square signal as test signal
Amplitude	the amplitude of the test signal (7.62283 mV fix)
Offset	the offset of the test signal (-7.62283 mV fix)
Frequency	specify the frequency of the test signal

Specify Channel Settings

Bipolar	perform a bipolar derivation between 2 input channels
Bandpass	perform a digital bandpass filtering of the input channels
Notch	perform a bandstop filtering to suppress the power line frequency of 50 Hz or 60 Hz
Load	allows loading a previously saved amplifier configuration
Save	allows saving the current amplifier configuration

Perform the following steps:

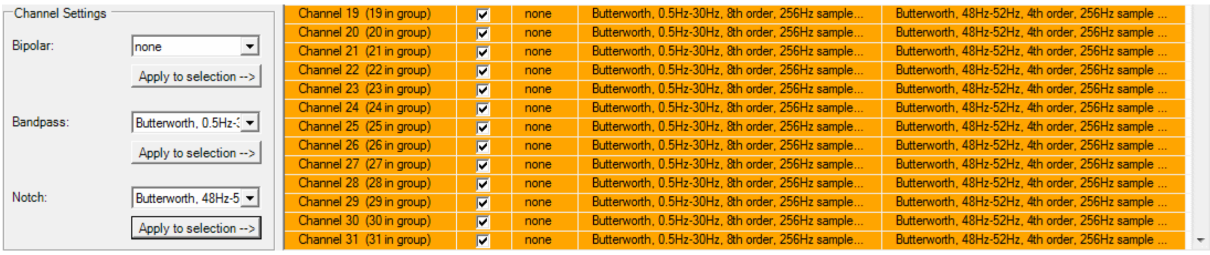
1. Select the channels in the listbox that should be edited. Use the Ctrl key or the Shift key to select multiple input channels.

2. Select the **Bandpass** filter `Butterworth, 0.5Hz – 30Hz, 8th Order, 256Hz Samplerate` and press the **Apply To selection** button to assign the bandpass to the specific channel. The selection is shown in the listbox.

3. Select the `Butterworth, 48Hz – 52Hz, 4th Order, 256Hz Samplerate` **Notch** filter to suppress the power line interference at 50 Hz and then press the **Apply to selection** button

4. To perform a bipolar derivation between channels 1 and 2 select the first channel in the listbox. Then select channel `2` under **Bipolar** and press the **Apply to selection** button. The settings appear in the listbox. This configuration subtracts channel 2 from channel 1 and the bipolar derivation will be visible on channel 1.

   NOTE: Select `none` under **Bipolar** and assign it to the channel if no bipolar derivation should be performed



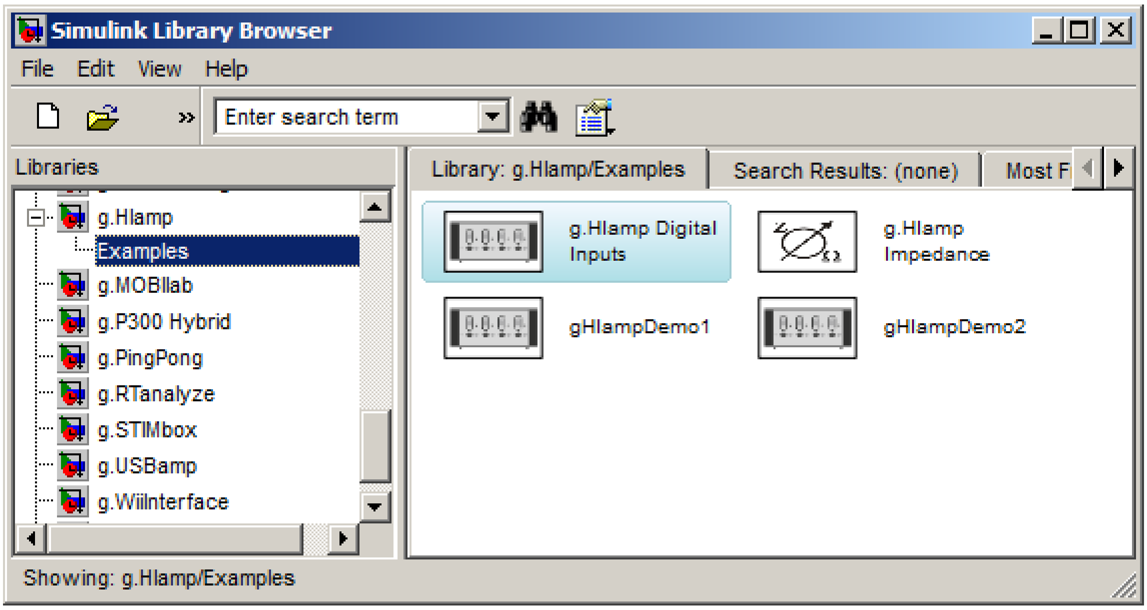
Press **OK** to close the window.

Running g.HIamp Highspeed

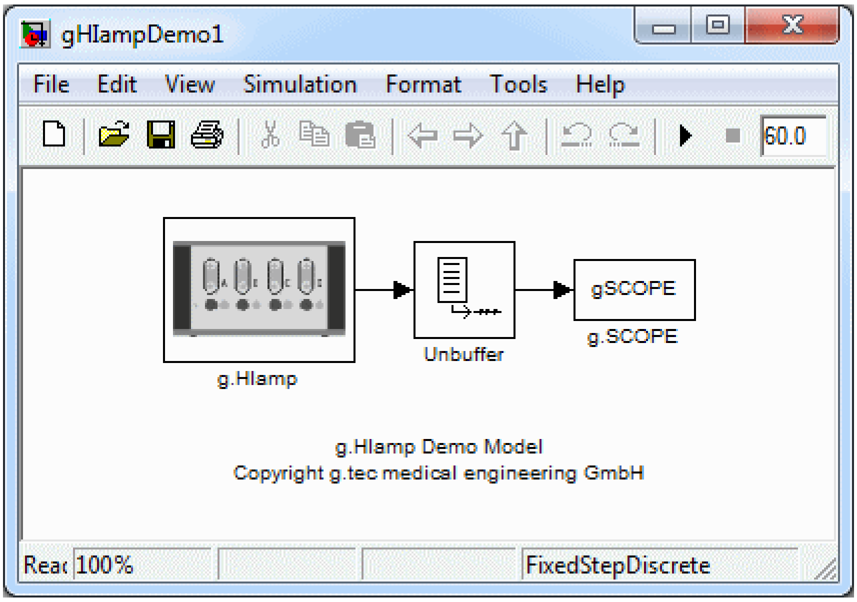
To test the g.HIamp Highspeed configuration on your system please perform the following example:

1. Start the MATLAB command window. See your MATLAB documentation if you are not sure how to do this.

2. Open the Simulink model by selecting **gHIampDemo1** from the **Examples** subsection of the **g.HIamp** entry in the **Simulink Library Browser**



or by typing `gHIampDemo1` into the MATLAB command window. The following window is opened:

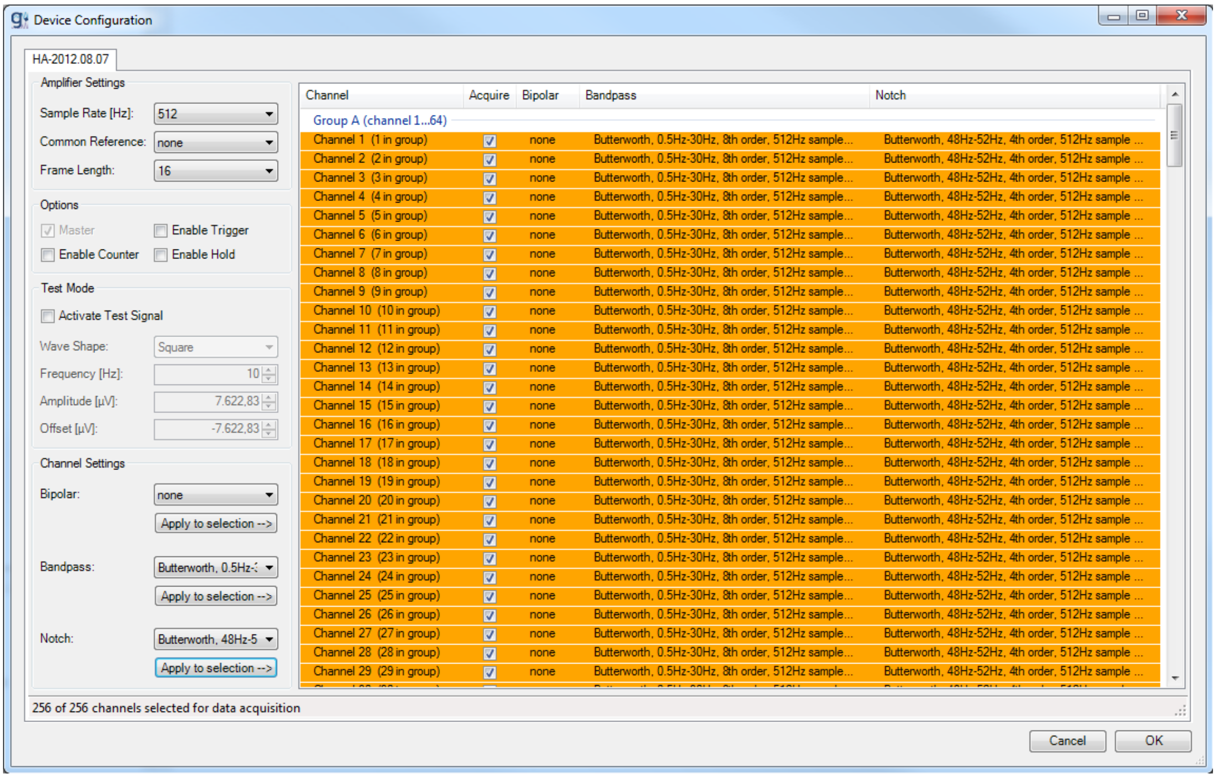


The Simulink model contains a **g.HIamp** block which reads in the data from the amplifier over USB. This example will acquire 80 channels at 512 Hz sampling rate.

3. Switch g.HIamp on and connect it to one free USB port. The power LED on g.HIamp must be on.

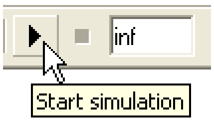
4. Connect a sine wave generator to channel 1 of g.HIamp. The sine wave should have an amplitude of ± 100 µV. This can be done e.g. with g.SIGgen.

5. Double click on the **g.HIamp** The **Device Configuration** dialog should look like this:



6. Click **OK**.

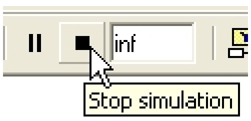
7. Start the model by pressing the **Start** button in the Simulink model



8. To view the signals double click on the **g.Scope** block. Please see g.tec Highspeed Library Description for more information on g.Scope block.

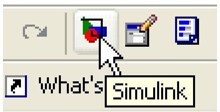
   Note that g.HIamp Highspeed reads in all biosignal data in µV

9. Stop the model with the **Stop** button and close it



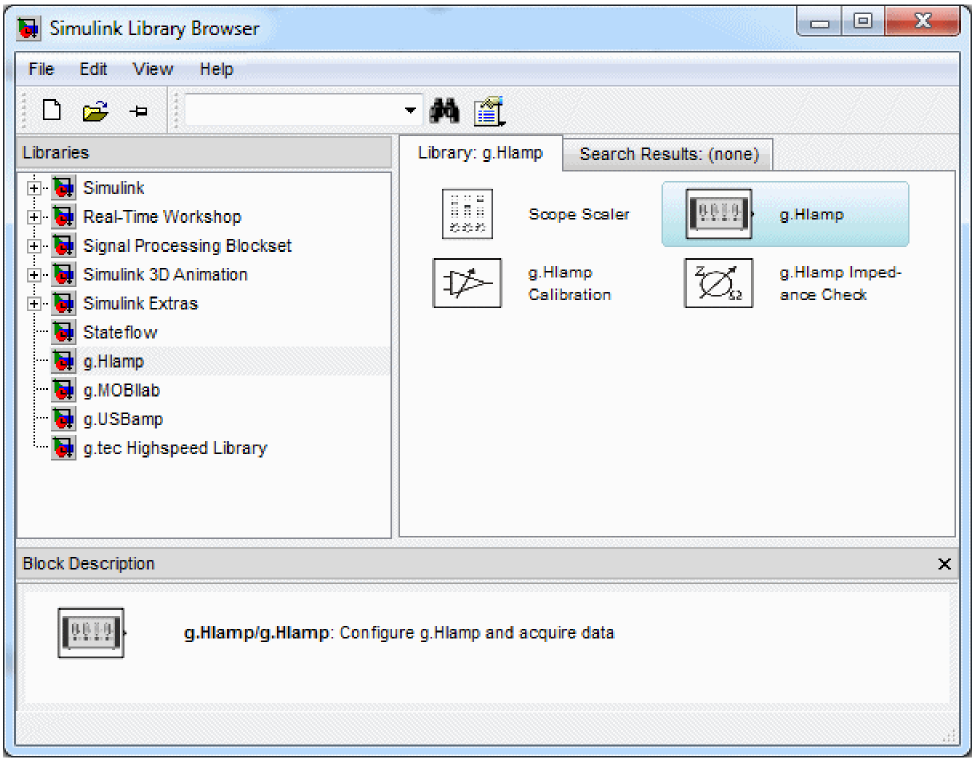
Creating a Simulink Model for Biosignal Acquisition

1. To create a new Simulink model click on the **Simulink** icon in the MATLAB window



or enter `simulink` into the MATLAB command window.

The **Simulink Library Browser** opens:

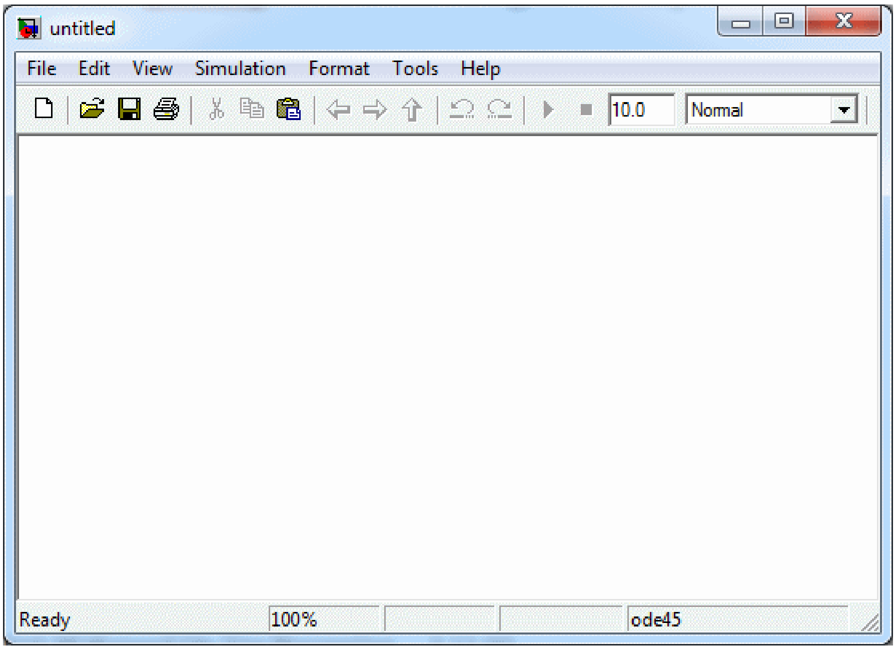


The **Simulink Library Browser** gives access to all Simulink based blocksets.

2. Scroll down to **g.HIamp** to show the biosignal data acquisition block.

3. Press the **Create a new model** icon in the **Simulink Library Browser**

to open an empty Simulink model:



4. Click on the **g.HIamp** block in the **Simulink Library Browser** and drag it into the new Simulink model

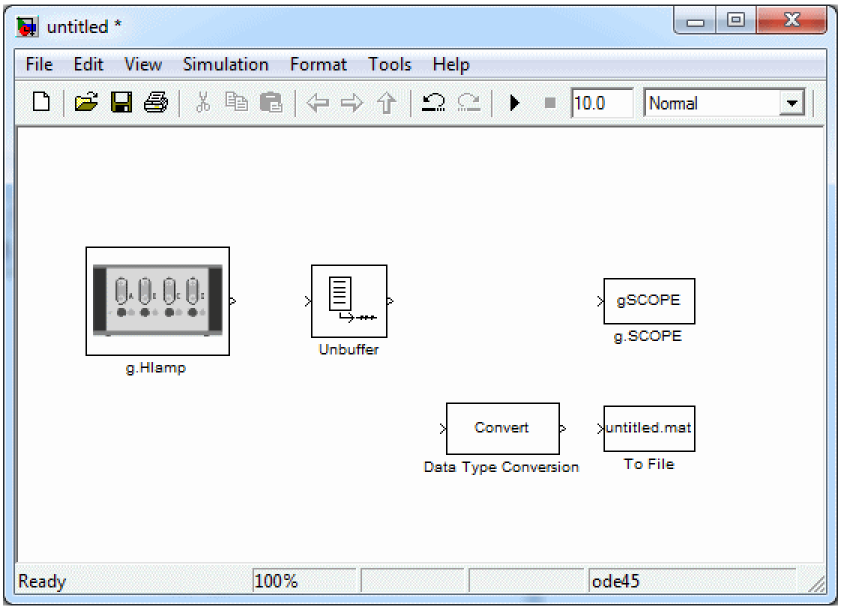
5. Open the **Signal Management\Buffers** path under **Signal Processing Blockset** in the **Simulink Library Browser** and copy the **Unbuffer** block into the new model

6. From the **g.tec Highspeed Library** copy the **g.Scope** block

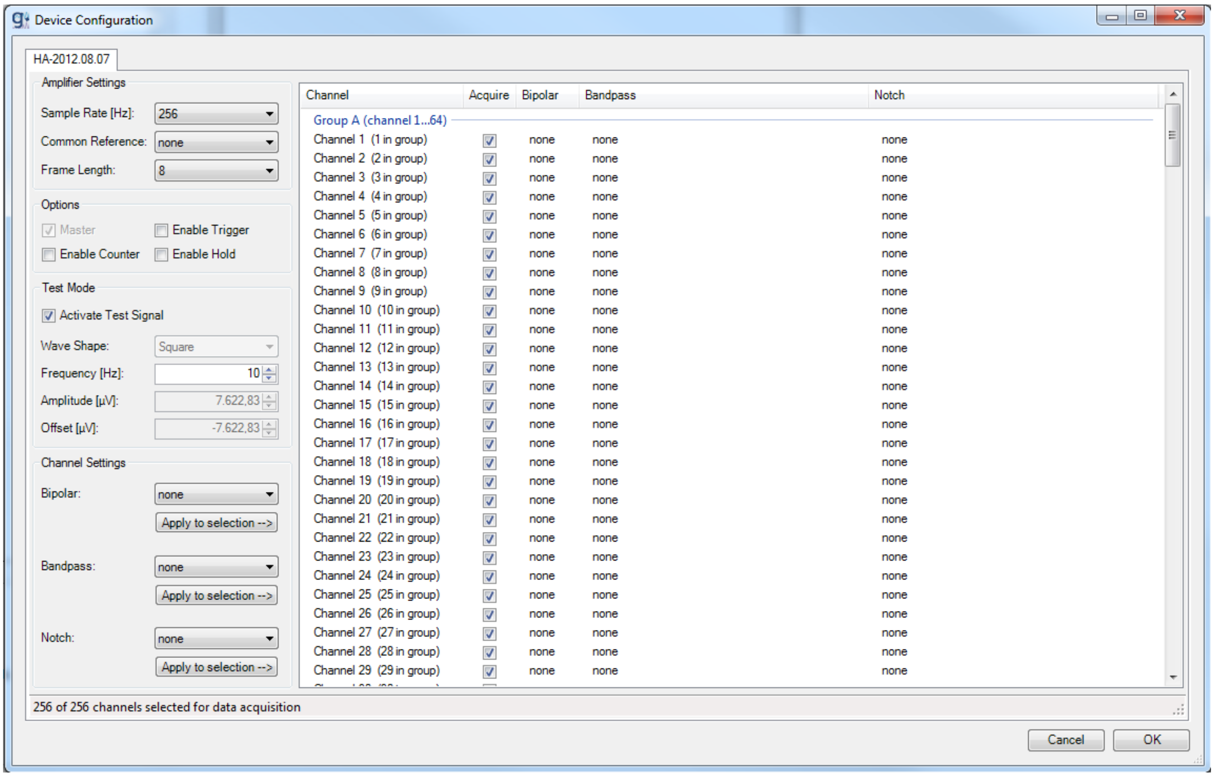
7. From **Sinks** under **Simulink** copy the **To File** block

8. From **Signal Attributes** under **Simulink** copy the **Data Type Conversion** block

9. Now your model should look like this:

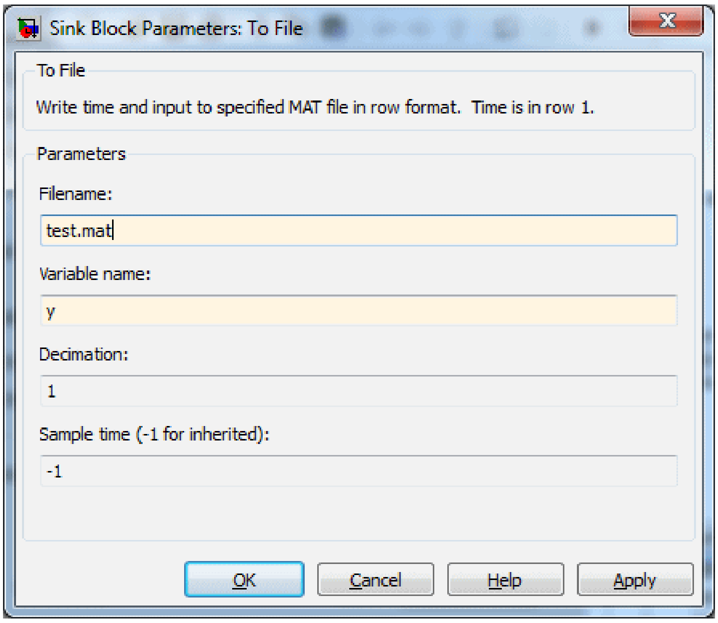


10. Double click on the **g.HIamp** block.

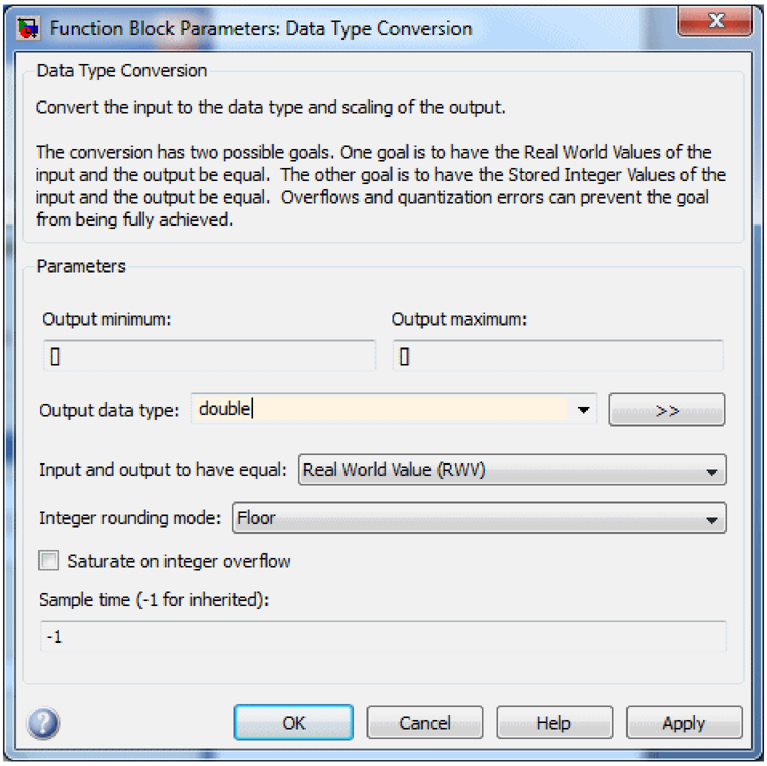


This example records the internal test signal on all available analog channels of g.HIamp. Therefore all analog channels have to be short cut to ground.

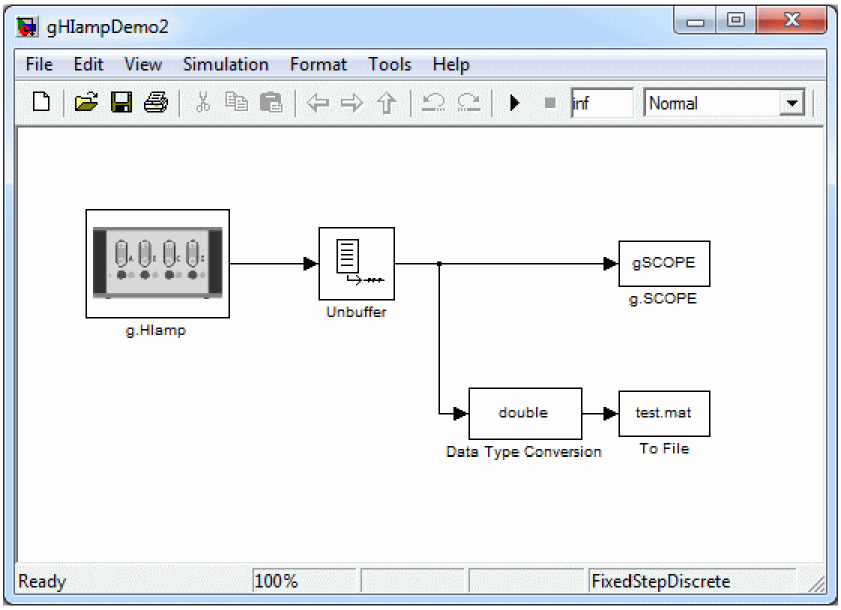
11. Double click on the **To File** block and enter the filename `test.mat` to store the data into variable `y`.



12. Double click on the **Data Type Conversion** block and change the **Output data type mode** to `double` because **g.HIamp** is acquiring the data as float32.



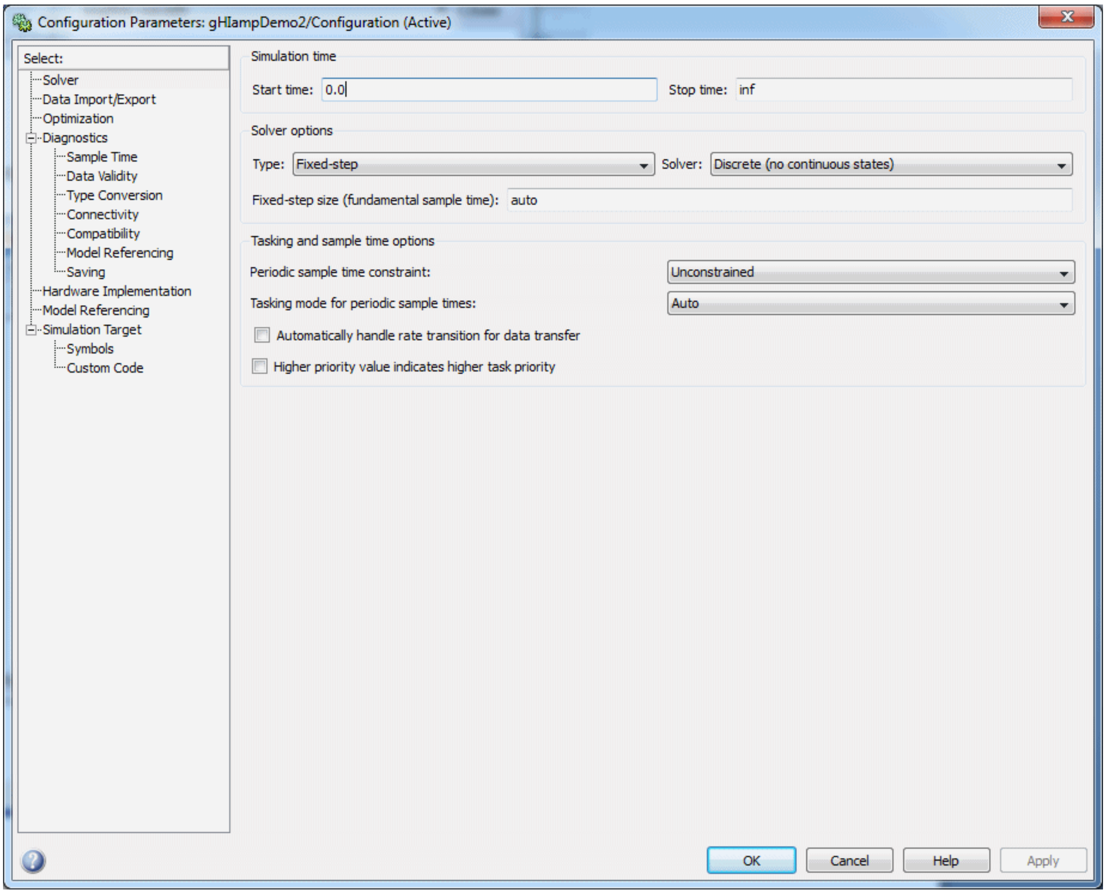
13. After configuring each block perform the connections as shown below:



Compare your model to the gHIampDemo2 model which can be opened from the **g.HIamp/Examples** subsection of the **Simulink Library browser** by double clicking the corresponding icon.

14. Click on the **Simulation** menu and select **Configuration Parameters**

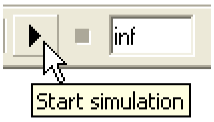
15. Set the **Stop time** to `inf` and the **Type** under **Solver options** to **Fixed-step**. **Solver** is set to **Discrete (no continuous states)**. The **Fixed-step size** is set to `auto` because **g.HIamp** block specifies the sample rate.



16. Confirm the settings and close the window with **OK**

17. Switch on g.HIamp and connect it to one free USB port. The power LED on g.HIamp must be on.

18. Start the model

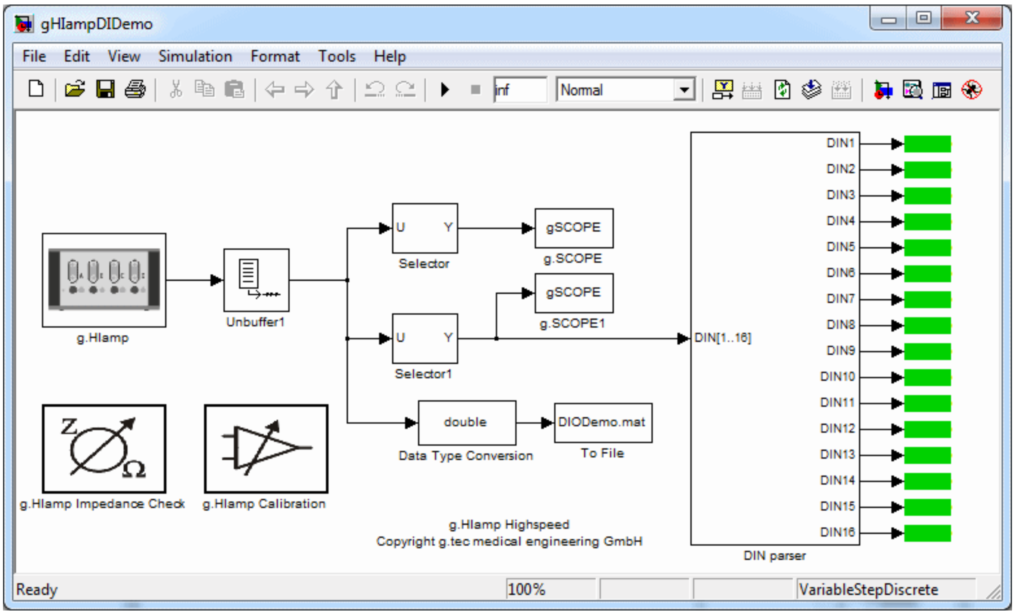


Using digital Inputs of g.HIamp

g.HIamp provides 16 digital inputs, the following example will show how these digital inputs are recorded in g.HIamp Simulink Highspeed.

NOTE: To follow this example E-Prime and a trigger cable for g.HIamp to the parallel port are required.

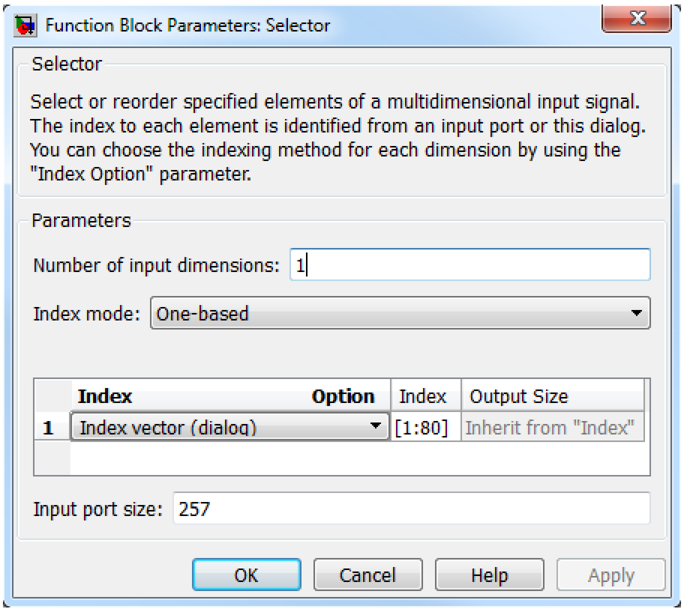
1. Open the model by double clicking the **g.HIamp Digial Inputs** icon in the **g.HIamp/Examples** section of the **Simulink Library Browser** or by typing `gHIampDIDemo.mdl` into the MATLAB commend window. This opens the following model:



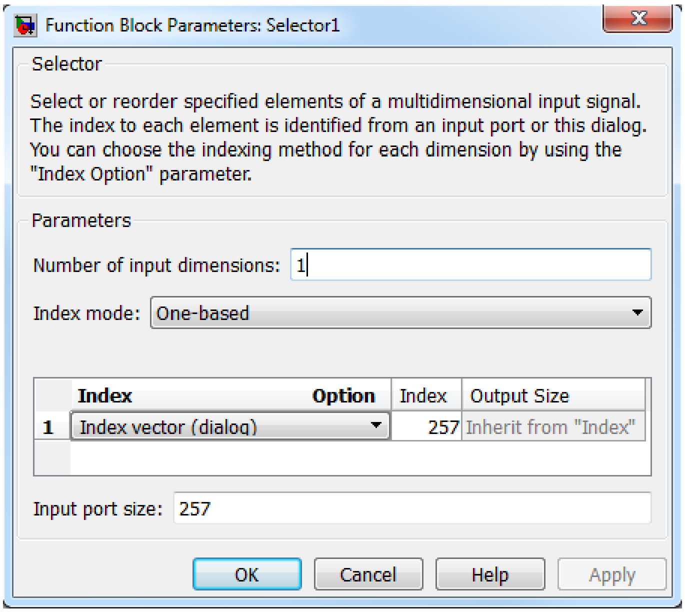
   This example will record 80 channels and the digital inputs of g.HIamp.

2. g.HIamp records digital inputs as single channel additionally to the currently selected analog channels. Values of this channel are ranging from 0 to 65535 corresponding to the 16 digital inputs fed to g.HIamp either on **DIGITAL IN 1** or **DIGITAL IN 2**.

3. The **Selector** block has to be set according to the number of channels of your g.HIamp. Default setting is for 256 channel version, therefore **Input port size:** is set to `257`. **Index** is set to `[1:80]` to extract first 80 channels of acquired data.



4. Similar settings apply to **Selector1** block, but **Index** has to be set to either 81, 145 or 257 corresponding to the maximum number of analog channels.



5. Start the E-Prime software with the Basic programmed part where the digital outputs of the parallel port are toggled to see the values recorded by g.HIamp.

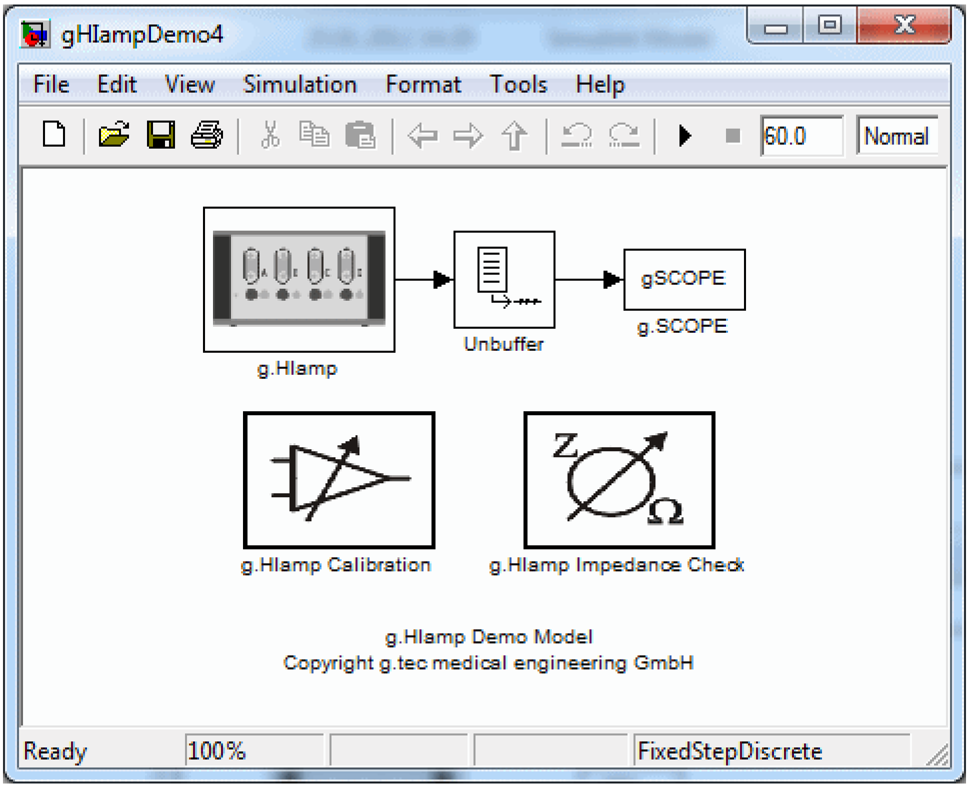
6. **g.Scope1** shows the current value of the trigger channel, **DIN parser** splits the 16 digital inputs.

   The current state of each digital input is indicated either in grey for low or green for high.

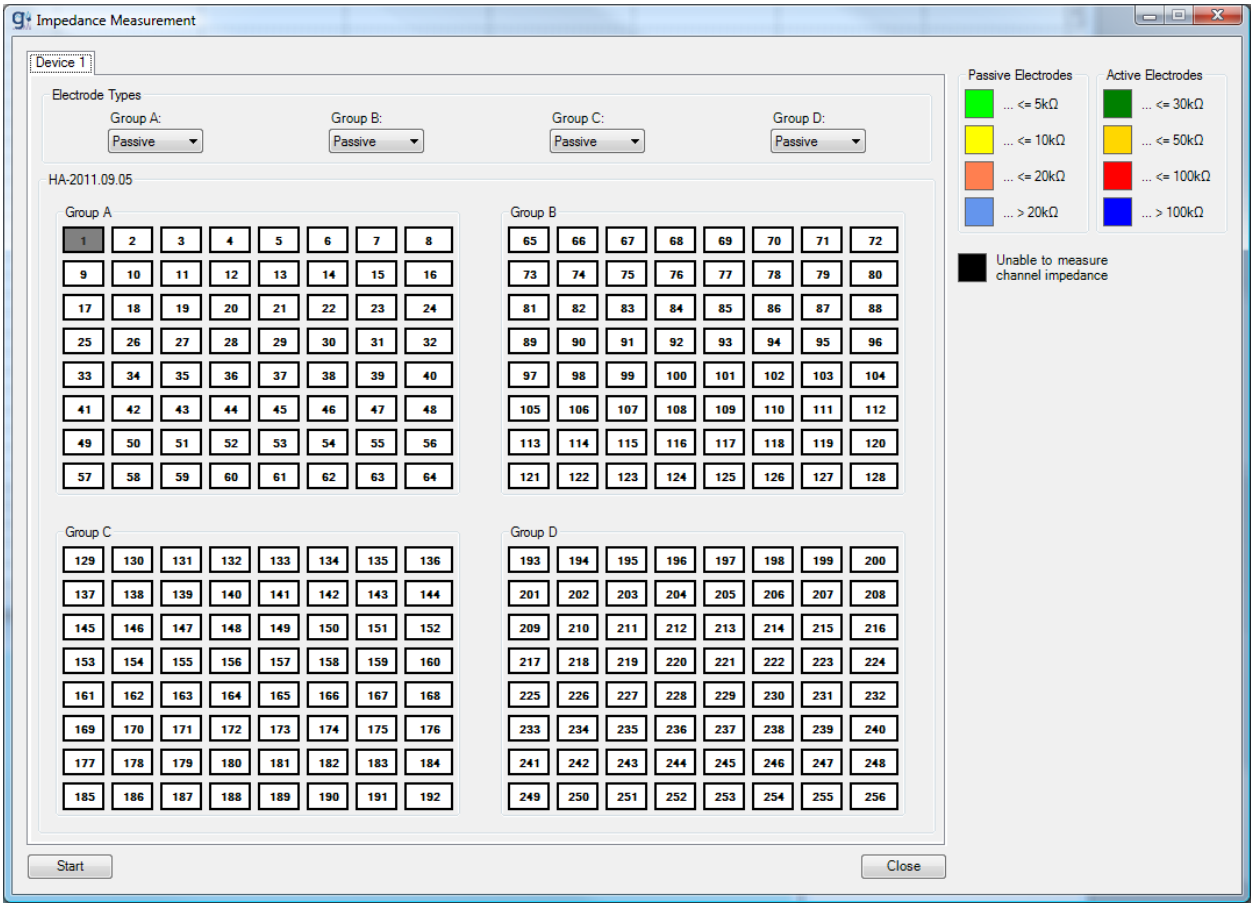
Impedance Measurement

g.HIamp has a build-in impedance check unit. To measure the impedances of active and passive electrodes perform the following steps:

1. Start the Simulink model by double clicking the **g.HIamp Impedance Check** icon in the **g.HIamp/Examples** section of the **Simulink Library Browser** or by typing `gHIampdemo4` at the MATLAB command line to open the model below:



2. Double Click **Impedance Check** block. The following window will appear:



The system creates a panel for each amplifier in the current model. The channels are labeled **1** to **256**. If the corresponding channel is selected the electrode impedance is measured.

3. Press **Start** to start the measurement. Impedance values are color coded for each channel. The colors indicate different ranges of the impedance values for active and passive electrodes, see legend on top right corner of the dialog.

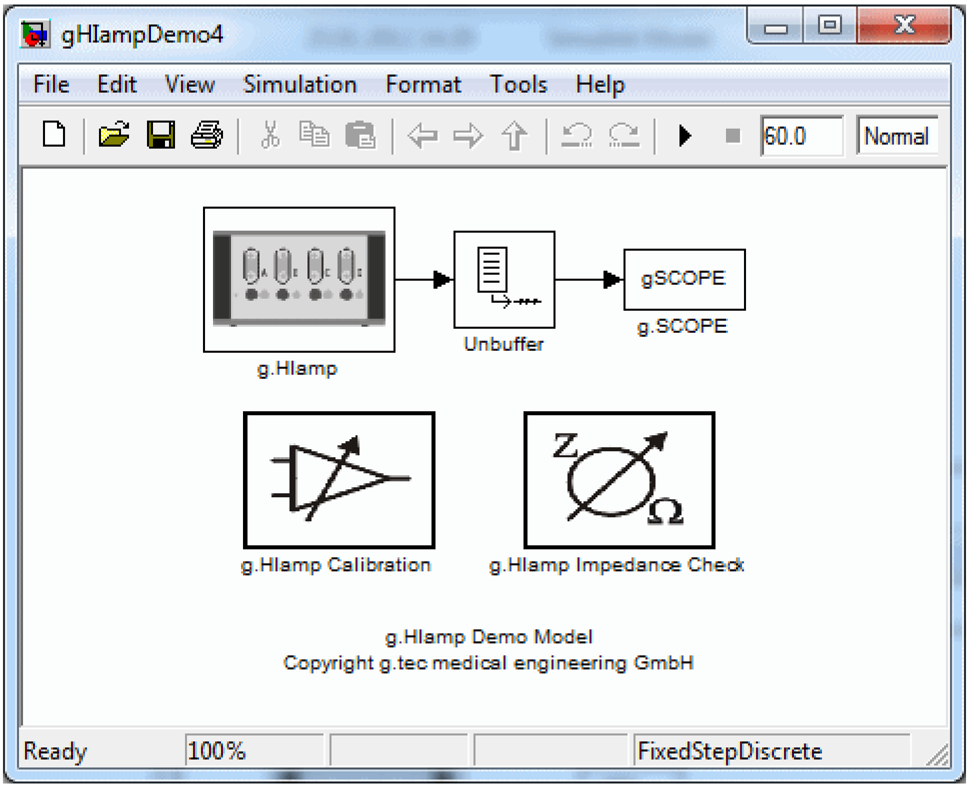
4. To stop the impedance measurement press **Stop**.

Calibration

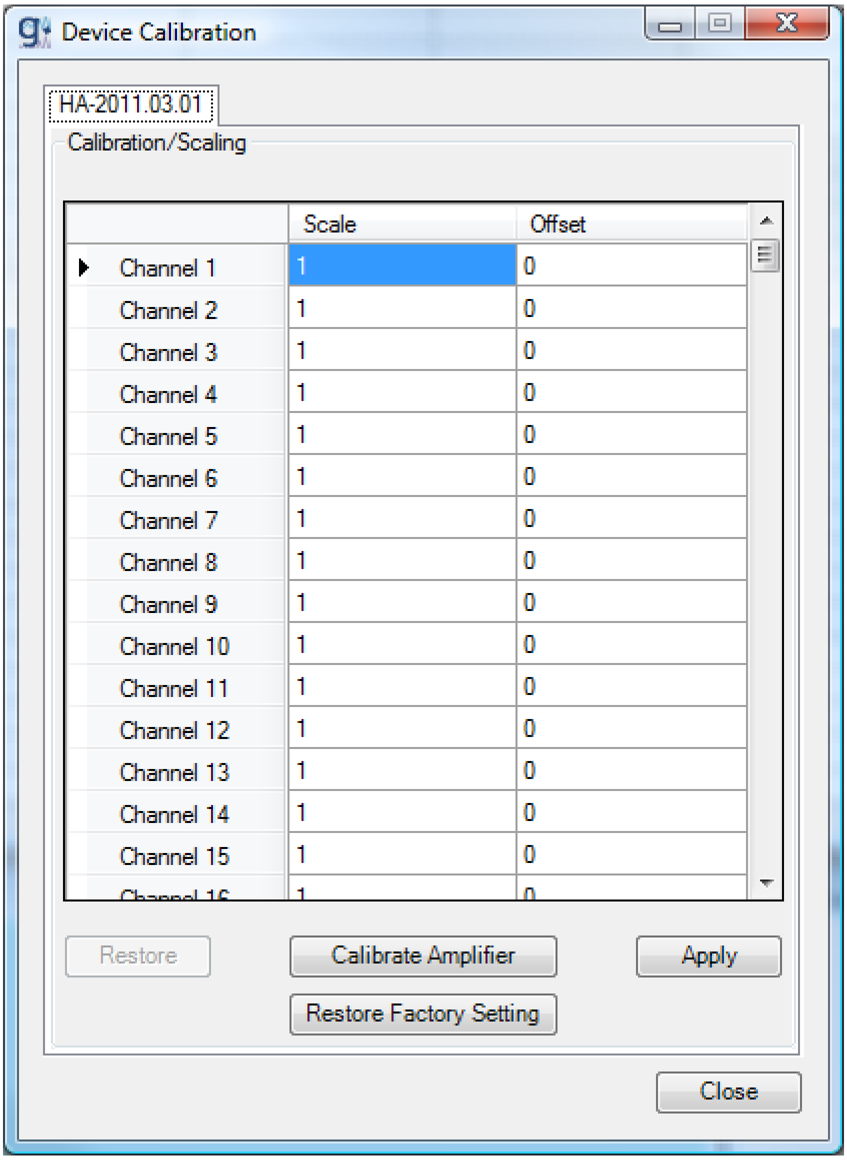
g.HIamp has a build-in calibration unit, all analog channels have to be short cut to ground to calibrate g.HIamp.

To calibrate g.HIamp perform the following steps:

1. Start the Simulink model `gHIampdemo4` from the MATLAB command line to open the model below:



2. Double click on the **Calibration** block. The following window will appear:



3. Press **Calibrate Ampifier** to perform the calibration and to get the **Offset** and **Scaling** values for each channel

4. Inspect the **Offset** and **Scaling** values. If you want to perform some changes use the editor boxes.

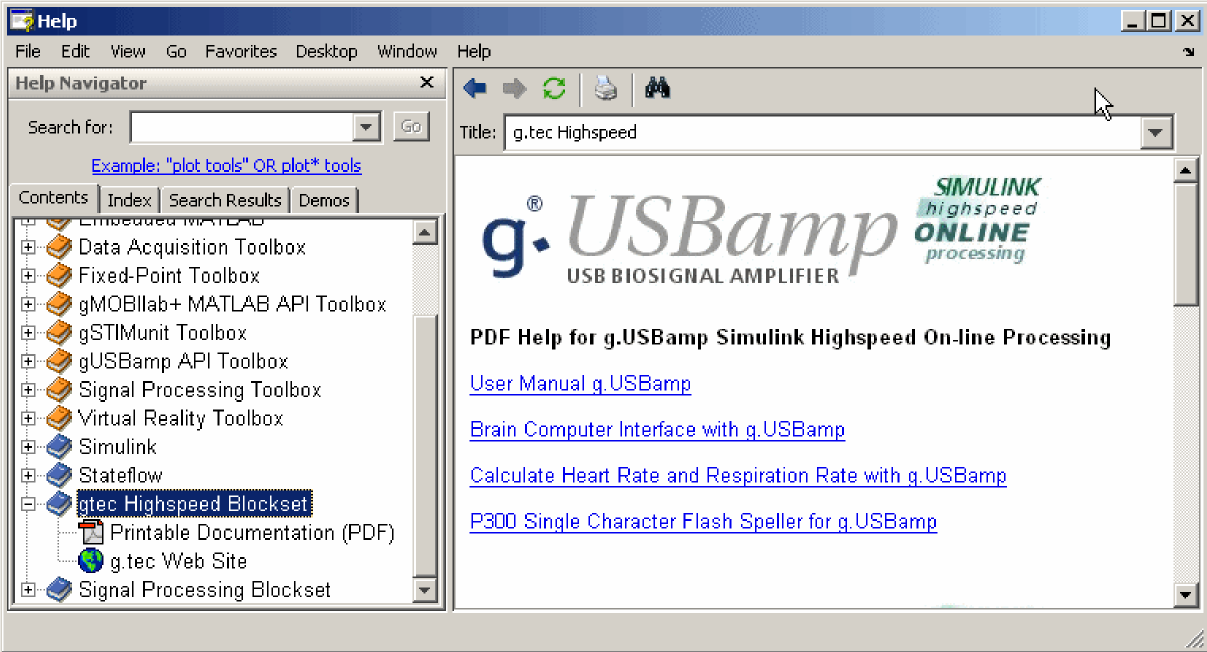
5. Press the **Apply** button to apply the calibration to g.HIamp

Help

g.tec Highspeed provides a printable documentation.

To access the help click on **MATLAB Help** in the **Help** menu of MATLAB. To access the help from command line type:

```
doc
```



The printable documentation is stored under

```
C:\Program Files\gtec\gtecHS\Help\gHIamp
```

as `gHIampHS.pdf`. Use Acrobat Reader to view the documentation.

Product Page

Please visit our homepage www.gtec.at for

- Update announcements
- Downloads
- Troubleshooting
- Additional demonstrations

GUGER
TECHNOLOGIES

contact information

g.tec medical engineering GmbH
Sierningstrasse 14
4521 Schiedlberg
Austria

tel. +43 7251 22240
fax. +43 7251 22240 39
web: www.gtec.at
e-mail: office@gtec.at