

# Appendix A

## ezAMI Development Environment Setup

### A.1 Installing Visual Studio 2019

ezAMI software requires MSVC C/C++ compiler. Installation of Visual Studio (VS) 2019 will automatically install the desired MSVC compiler VS2019 can be download from the link below

<https://visualstudio.microsoft.com/thank-you-downloading-visual-studio/?sku=Community&rel=16>

What you download from the link above is a VS2019 community version installer `vs_community__1176045115.1594347171.exe`. Launch the installer and follow the installation wizard to install the software on the C drive of Windows10 X64 version. If successful, the software installation location will be under Local Disk(C:)>Program Files(x86). Software folder will be look like in figure A.1.

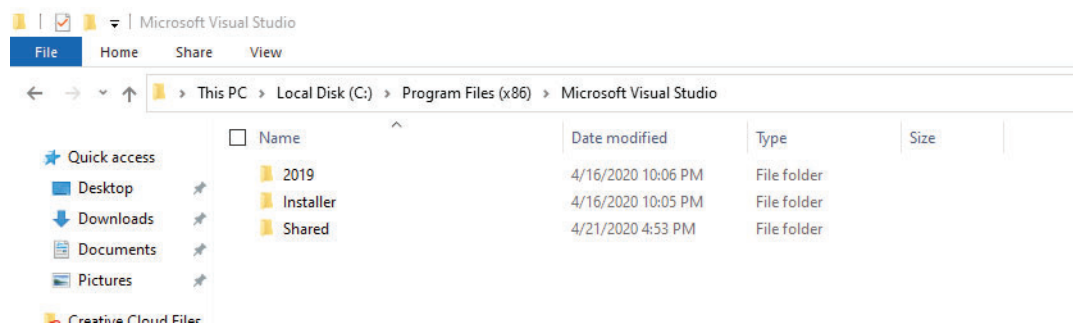


Figure A.1: VS 2019 installation snapshot

## A.2 Installing QT IDE 5.12

The ezAMI software is developed using C++ under QT IDE 5.12. The QT IDE 5.12 can be downloaded from the link below

<https://www.qt.io/offline-installers>

Download Qt 5.12.9 for Windows as shown in figure A.2

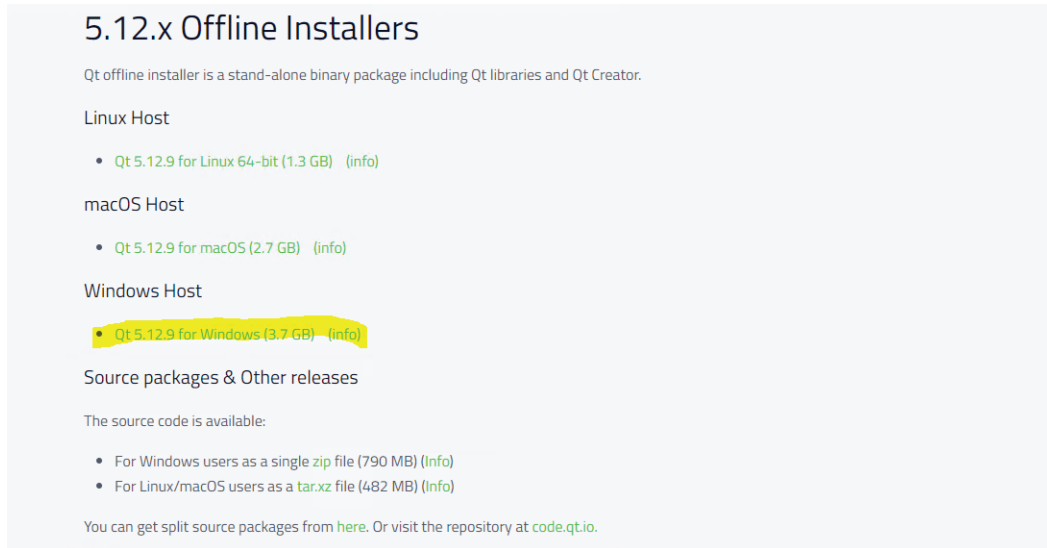
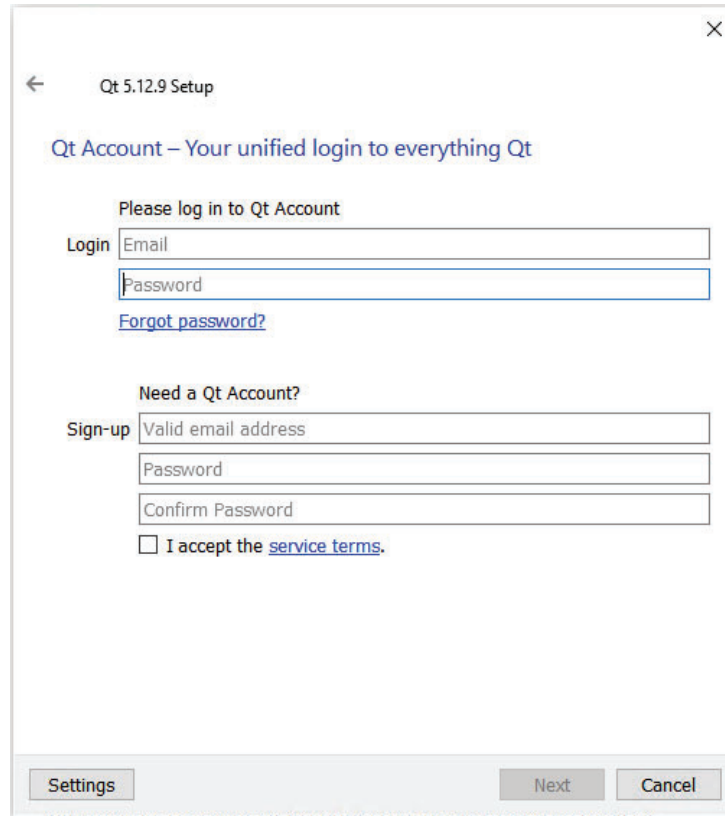


Figure A.2: QT IDE download

QT IDE installation requires a Qt account. If you have one already, you can just login to continue. Otherwise you have to sign one up (see figure A.3). Then you can follow the installation wizard till to the step **Qt 5.12.9 Setup**. Choose **Select All** as shown in figure A.4. After that, you can just follow the default setting to complete the installation. Once installation is completed, you can launch the Qt Creator 4.9 as shown in figure A.5. The main interface is shown in figure A.6.



Qt 5.12.9 Setup

Qt Account – Your unified login to everything Qt

Please log in to Qt Account

Login

Email

Password

[Forgot password?](#)

Need a Qt Account?

Sign-up

Valid email address

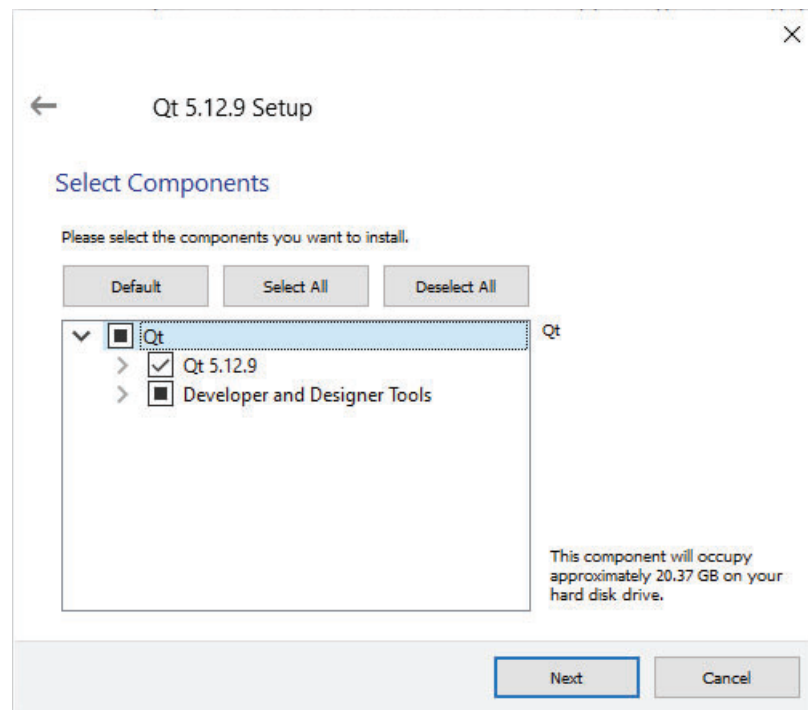
Password

Confirm Password

☐ I accept the [service terms](#).

Settings Next Cancel

Figure A.3: QT IDE installation login



Qt 5.12.9 Setup

Select Components

Please select the components you want to install.

Default Select All Deselect All

Qt

- ☒ Qt
  - ☒ Qt 5.12.9
  - ☒ Developer and Designer Tools

This component will occupy approximately 20.37 GB on your hard disk drive.

Next Cancel

Figure A.4: QT IDE installation setup

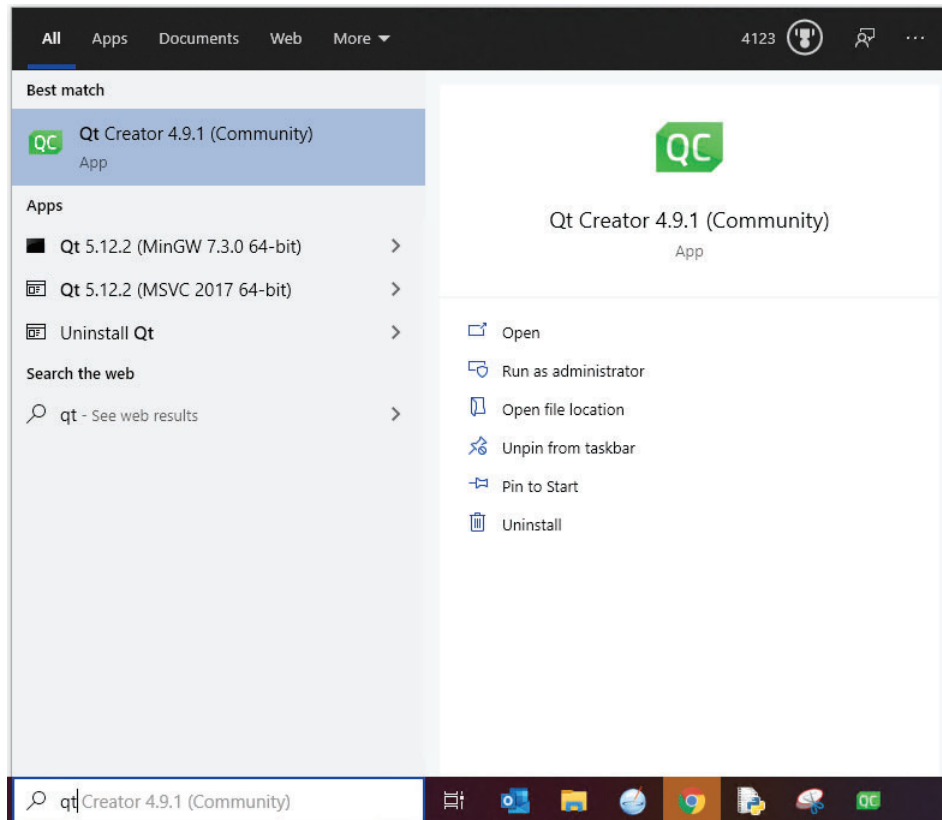


Figure A.5: Launch QT Creator

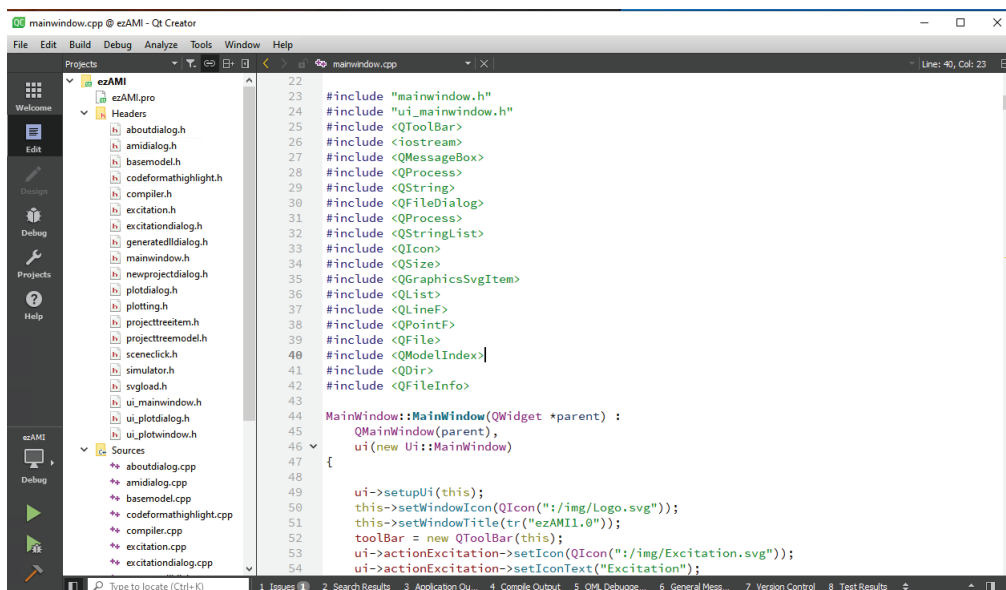


Figure A.6: QT Creator interface

## Appendix B

### Tutorial for LVFFN example in ezAMI

#### B.1 Install ezAMI Software

Download ezAMI installer from the link below

[www.ezamiuiuc.net](http://www.ezamiuiuc.net)

Launch installer and follow the installation wizard to complete the installation (figure B.1).

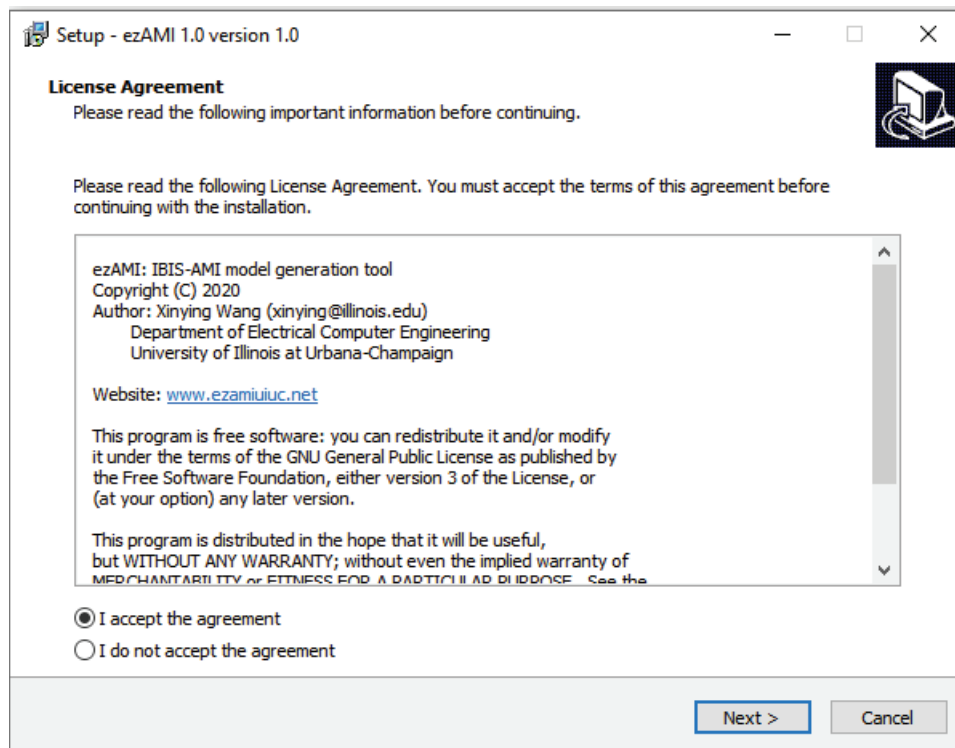


Figure B.1: ezAMI installation

## B.2 LVFFN example

Launch software and click the "AMI" icon pointed by the blue arrow to draw the schematic (figure B.2)

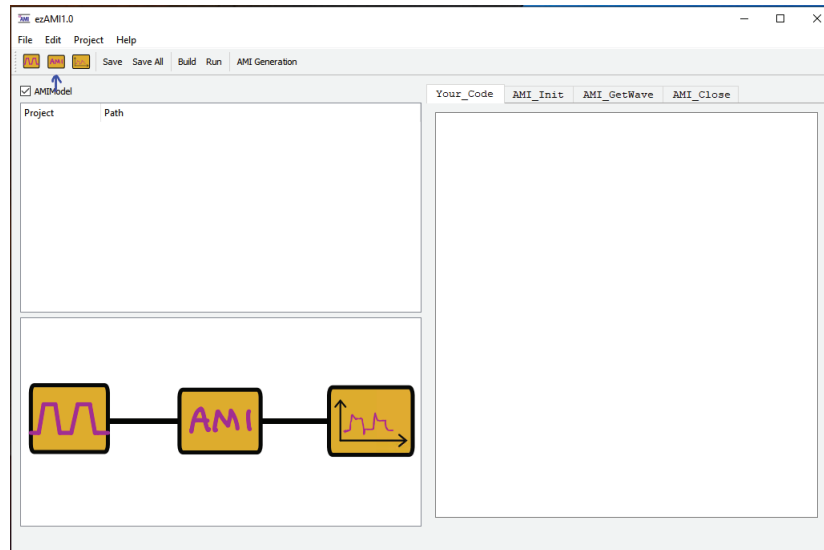


Figure B.2: Add schematic

Select "File→Example→LVFFN" as figure B.3 shows

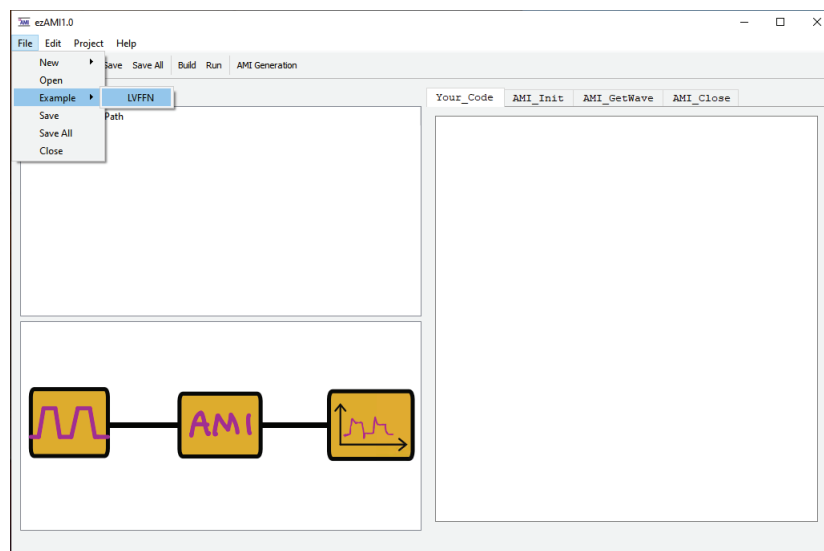


Figure B.3: Launch LVFFN example

A window pop up to let select or create a folder to store LVFFN project

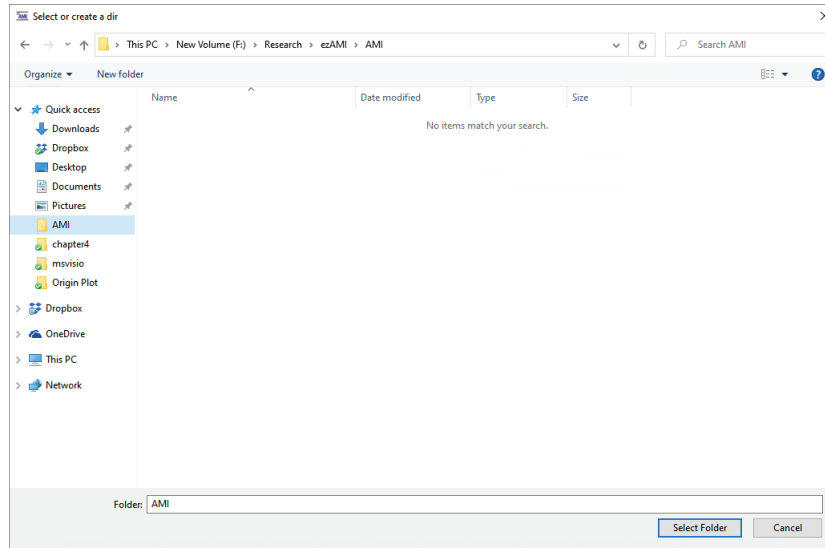


Figure B.4: Select folder to save LVFFN project

Once the LVFFN example project is loaded, the software main interface will be showing like in figure B.5

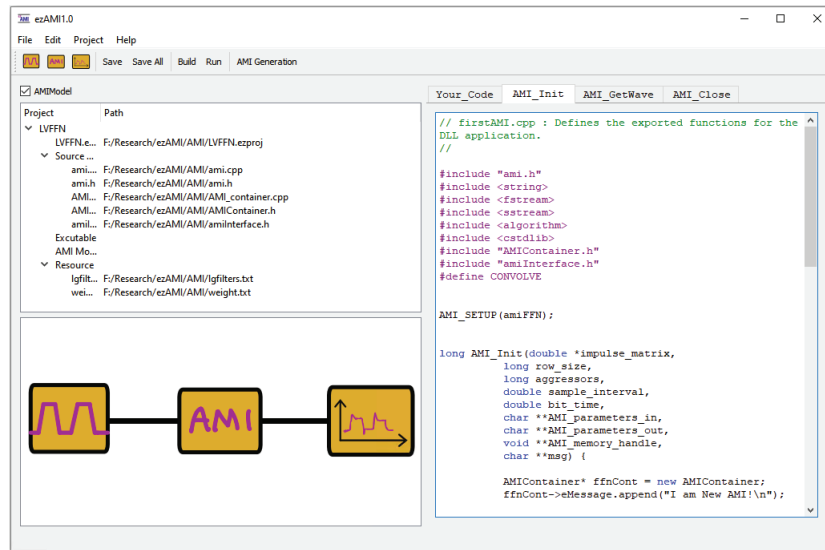


Figure B.5: Software main interface after LVFFN example is loaded

Click "Build" in menu bar or select from "Project→Build" to compile the code to generate the DLL file for simulation. If compilation is successful, it will look like in figure B.6).

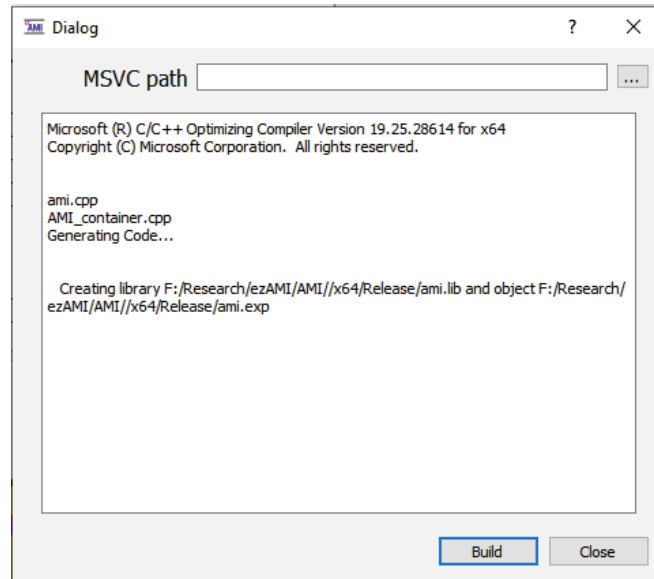


Figure B.6: Software main interface after LVFFN example is loaded

Double-click the "AMI" icon in the schematic window. A dialog window pop up to let you select the DLL file you just compiled (see figure B.7). The dll file will be in the project folder.

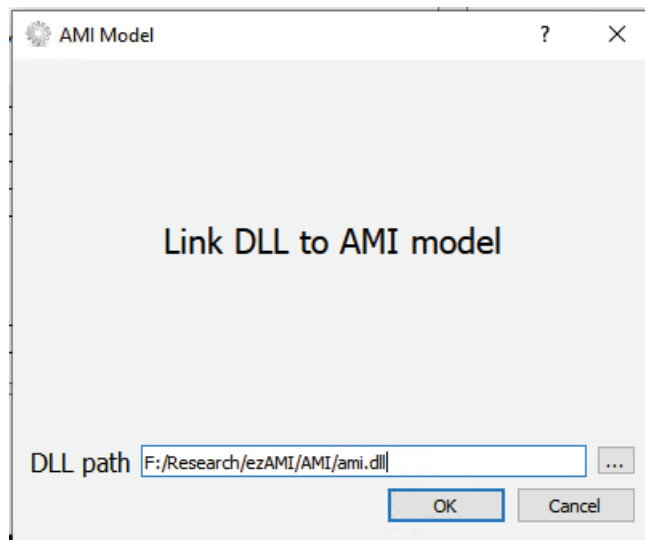


Figure B.7: Link dll file for simulation



Double-click the Excitation icon in schematic window. It is the first one from left. A dialog window is popped up to let you set the excitation setting. Choose PAM-4 and 100 bits as shown in figure B.8. Leave other settings as default, then click "Ok"

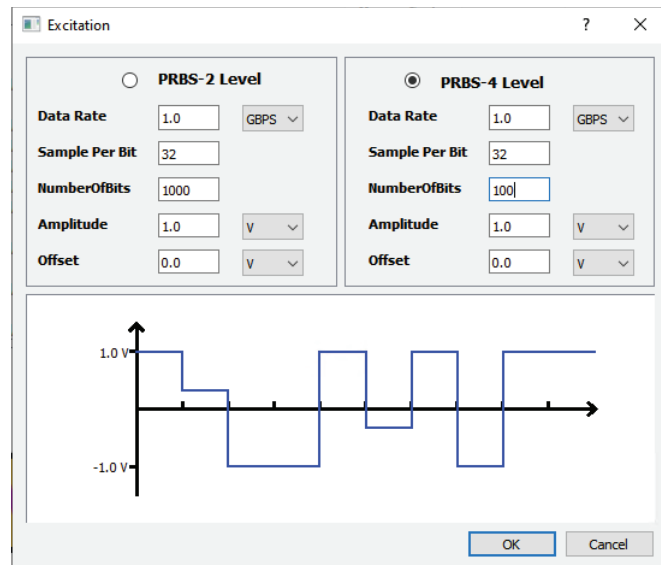


Figure B.8: Excitation generation

The next is to click "Run" in menu action bar or select "Project→Run" to launch simulation. If no errors happen, a window with plot for excitation wave and model output will be shown as figure B.9

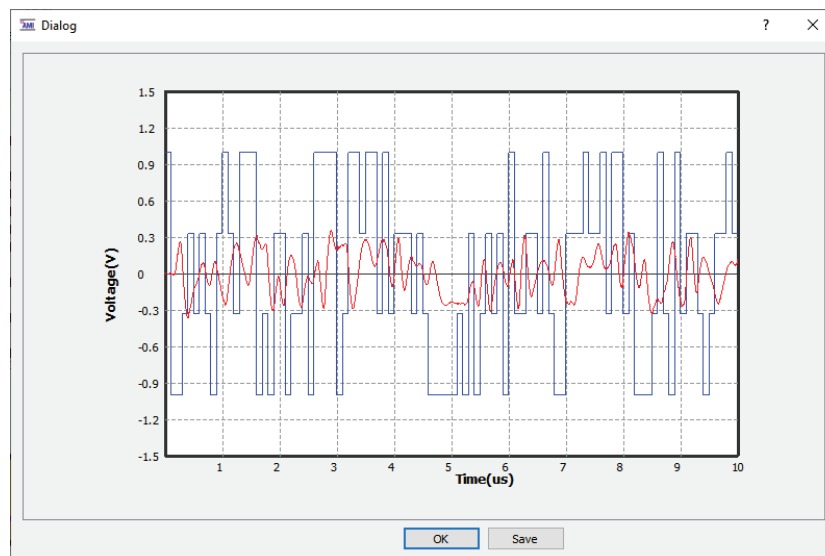


Figure B.9: Result display

The last step is to generate the AMI model for simulation in circuit simulator. Click "AMI Generation" action in menu action bar or select from "Project→AMI Generation". Select "X64". Leave ".ami Settings" as default since it is still being completed.

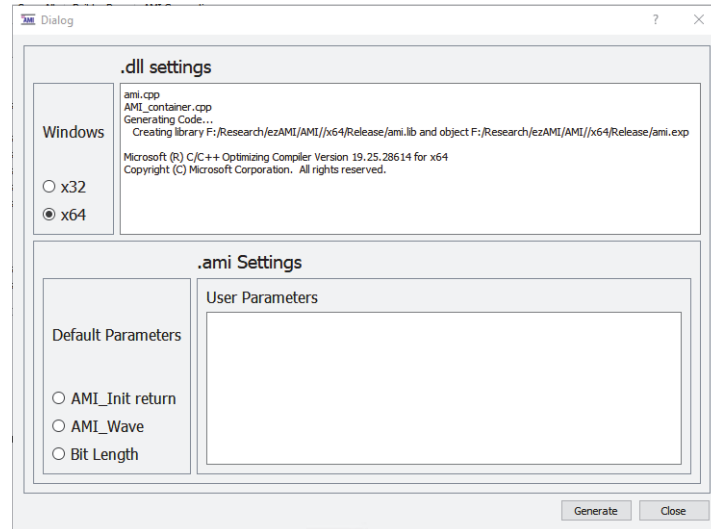


Figure B.10: Generate AMI model

If AMI model generation is successful, you will see both DLL file and .ami file appear in the project management region under the "AMI Model" node, as shown in figure B.11. Both files are the AMI model package you can use for simulation with commercial circuit simulator.

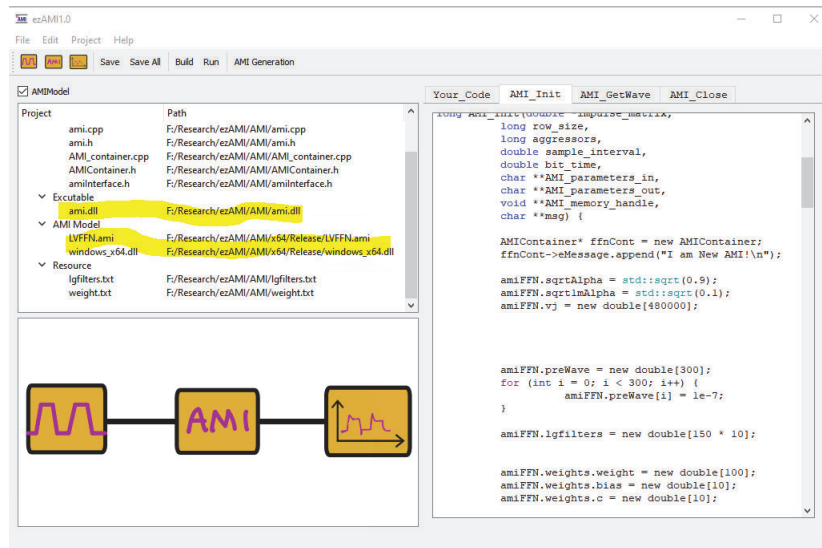


Figure B.11: Generated AMI model