ECSE 308:
Introduction to
Communication Systems and Networks

L0: Tutorials

Tutorial 1: Simulink

Tutorial 2: Wireshark

Tutorial 1:

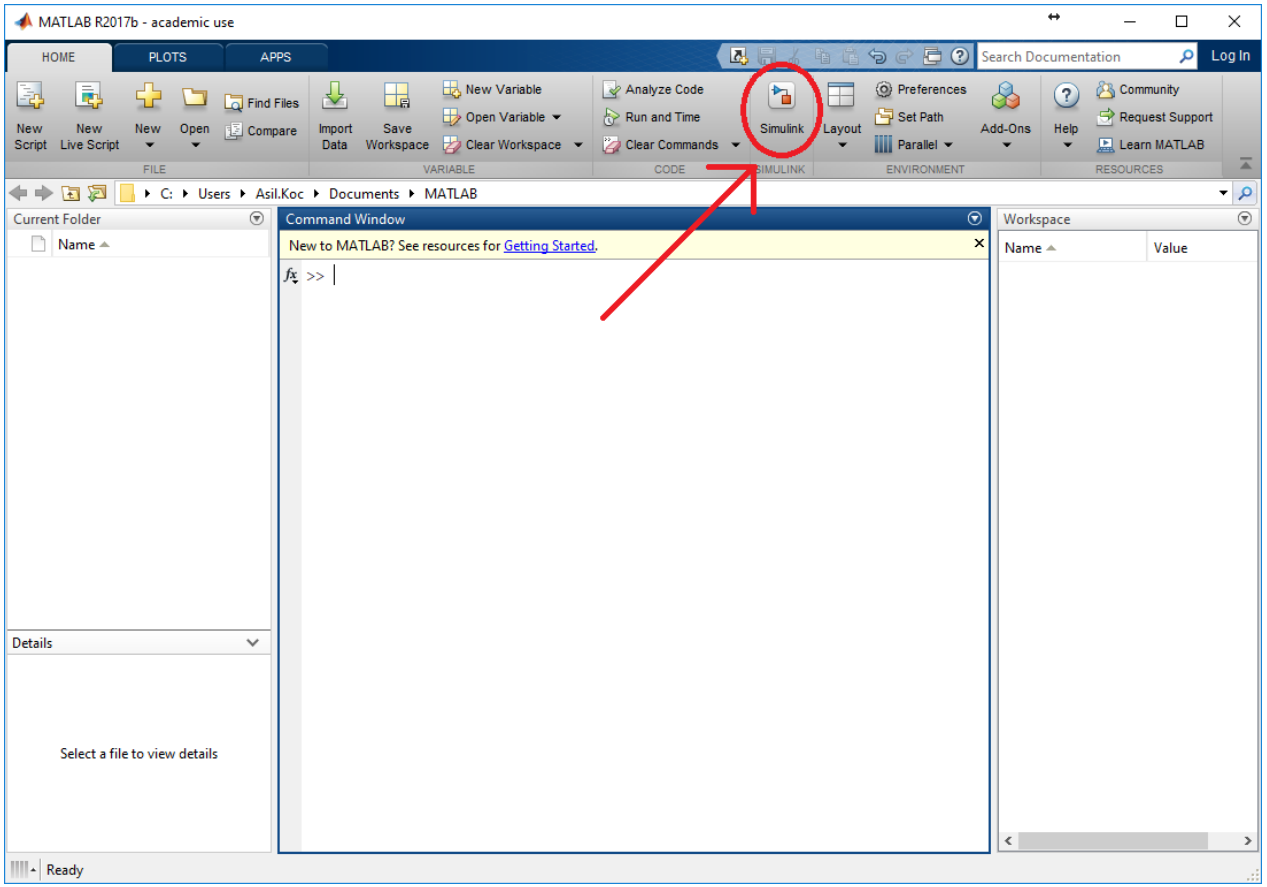
Simulink Tutorial

I. Introduction

- Simulink is an environment for simulation and model-based design for dynamic and embedded systems.
- It provides an interactive graphical environment and a customizable set of block libraries that let you design, simulate, implement, and test a variety of time-varying systems, including communications, controls, signal processing, video processing, and image processing.
- Simulink offers:
  - A quick way of develop your model in contrast to text based-programming language such as e.g., C.
  - Simulink has integrated solvers. In text based-programming language you need to write your own solver.

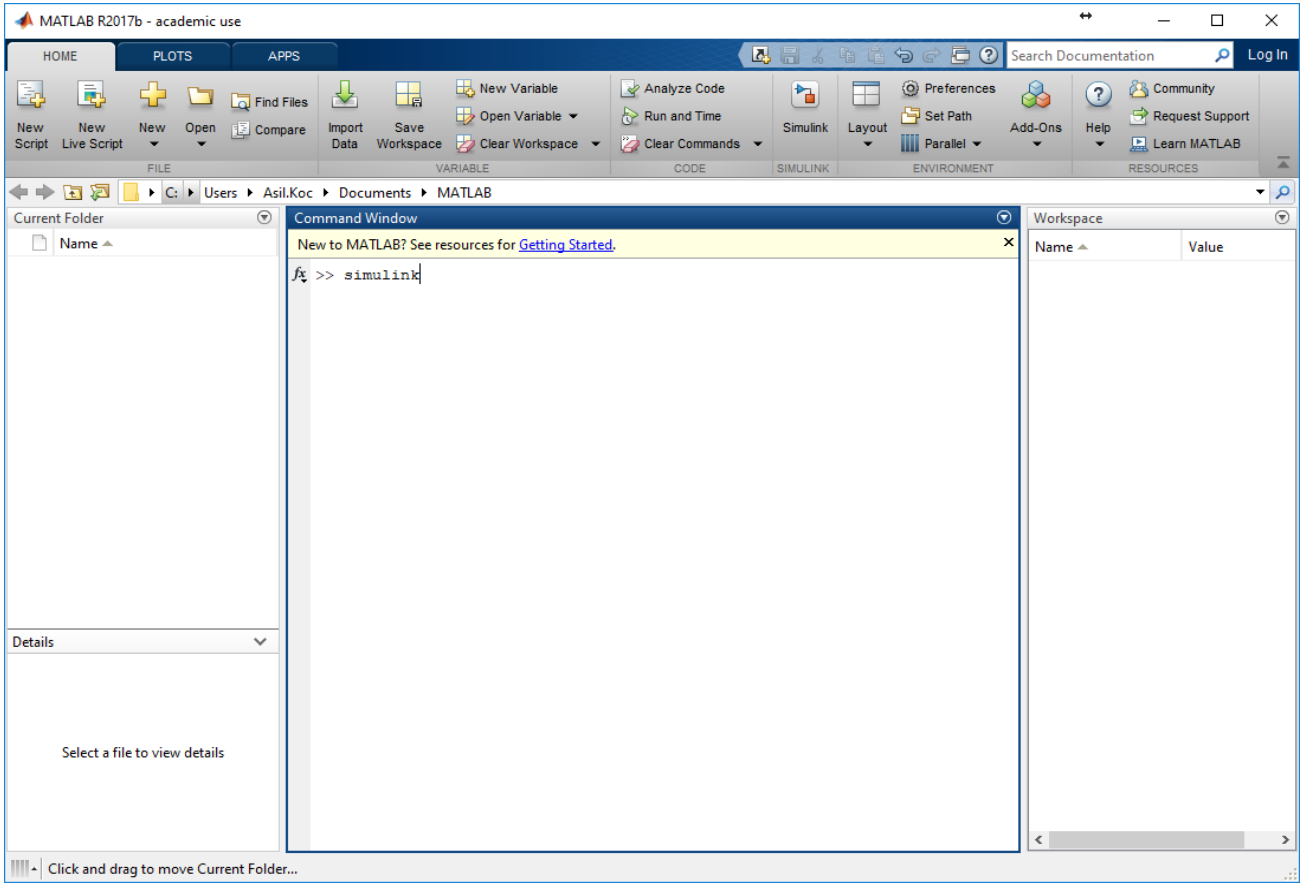
II. How to Open Simulink

- Open MATLAB and select the Simulink icon in the Toolbar:



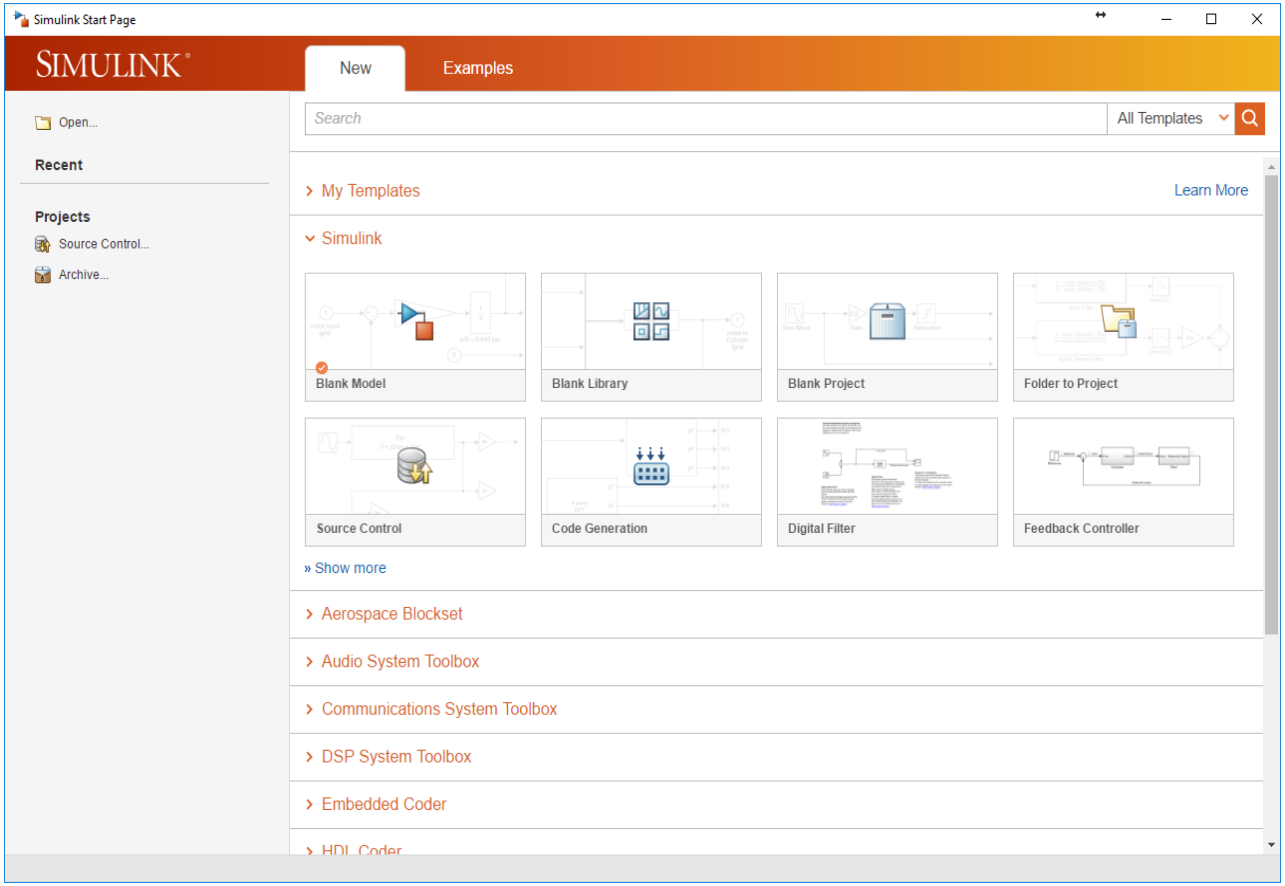
II. How to Open Simulink

- Or type "**simulink**" in the Command window, like this:



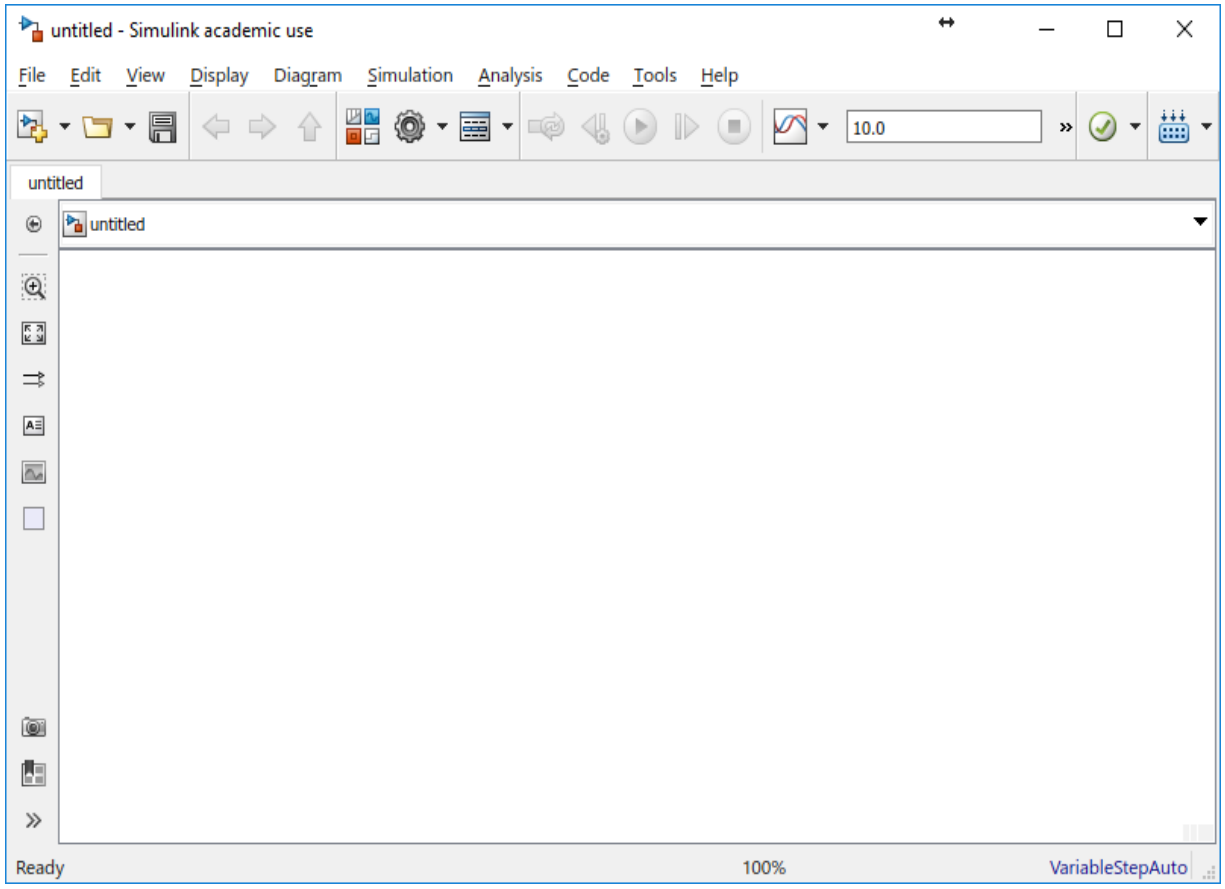
II. How to Open Simulink

- Then, the following window appears (Simulink Start Page):



II. How to Open Simulink

- Select "**Blank Model**" to open the following **Simulink Editor** window.

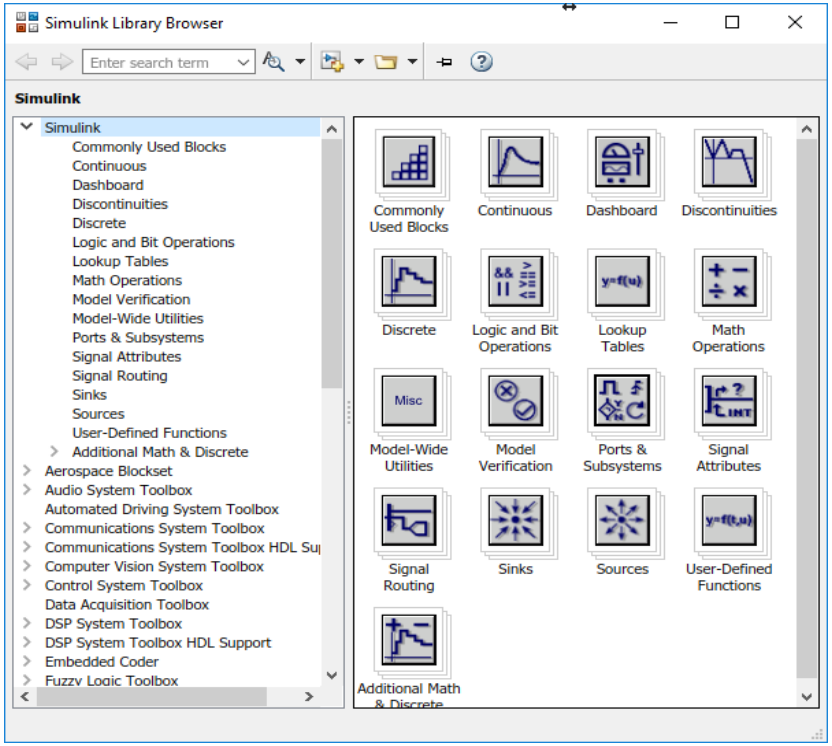


III. Create a Simple Model

- You can use Simulink to model a system and then simulate the dynamic behavior of that system.
- In this part, a simple model is created, which integrates a sine wave signal to a cosine signal and then displays the result, along with the original signal, in a scope window.
- The basic techniques used to create the simple model in this tutorial are the same techniques for more complex models.
- To create this simple model, you need four Simulink blocks:
  - **Sine Wave** — Generates an input signal for the model.
  - **Integrator** — Processes the input signal.
  - **Bus Creator** — Combines the input signal and processed signal into one signal.
  - **Scope** — Visualizes the signals.

III. a. Open the Simulink Library Browser

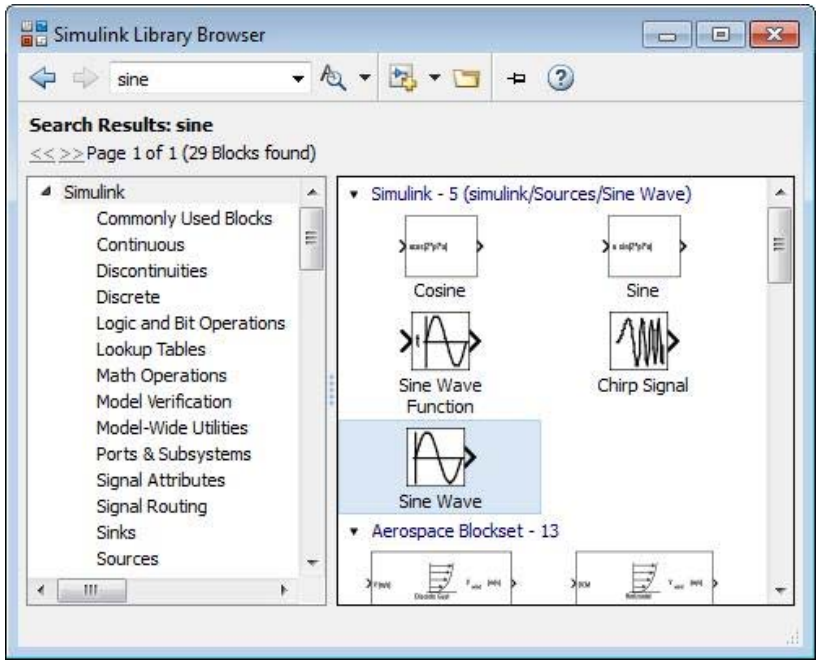
- In "**Blank Model**" window, click the **Library Browser** ( ) button.
  A short delay occurs for the first time opening the Simulink Library Browser.
- In order to design the Simulink models, you may now drag the blocks you want to use from the Simulink Library Browser to the model surface.



III. b. Search for Specific Blocks

You can search all the available block libraries.

- Search for a Sine Wave block. In the search box on the browser toolbar, enter **sine**, and then press the Enter key. Simulink searches the libraries for blocks with **sine** in their name, and then displays the blocks
- To get detailed information about a block, right-click a block and then select **Help for the <block name>**. The Help browser opens with the reference page for the block.
- To view block parameters, right-click a block and then select **Block Parameters**. The block parameters dialog box opens.

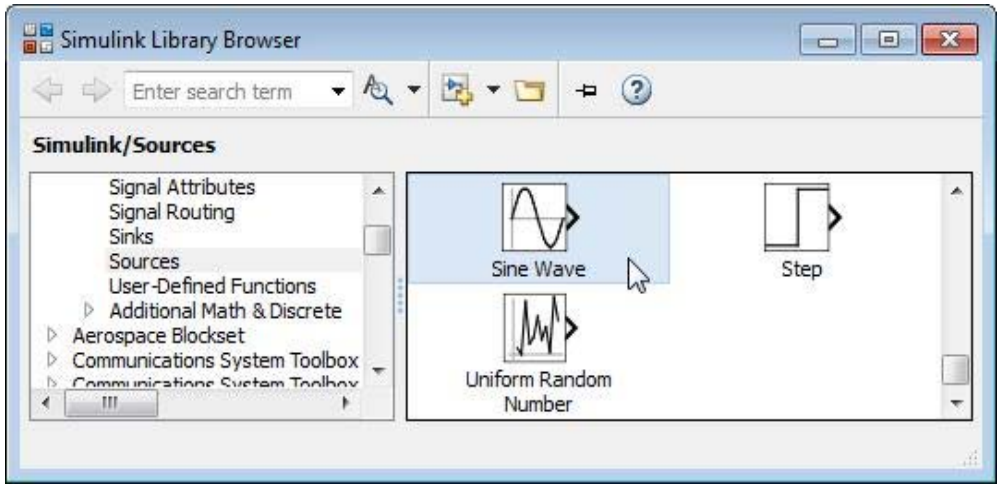


III. c. Add Blocks to a Model

You build models by dragging blocks from the Simulink Library Browser window to the Simulink Editor window or single-clicking your model and entering a search term. You can search all the available block libraries.

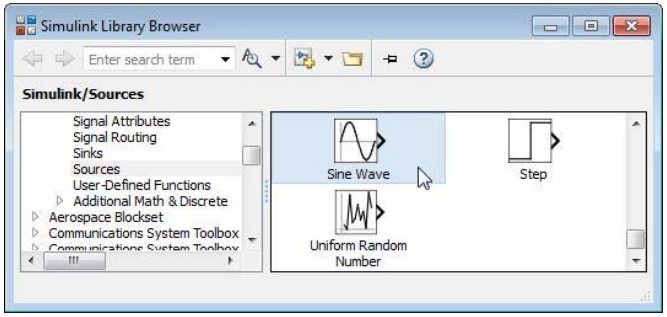
To build a model, begin by copying blocks from the Simulink Library Browser to the Simulink Editor.

- In the Simulink Library Browser, select the **Sources** library.
- From the right pane, select the Sine Wave block.

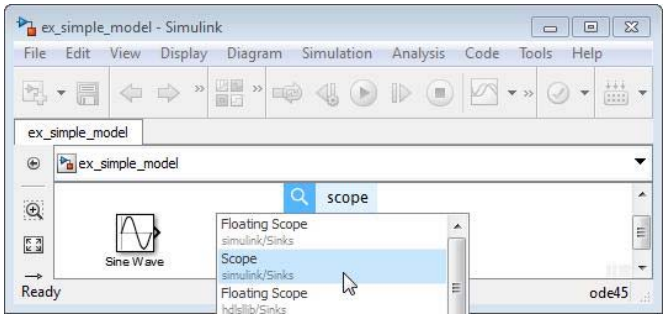


III. c. Add Blocks to a Model

- Drag the Sine Wave block to the Simulink Editor. A copy of the Sine Wave block appears in your model.

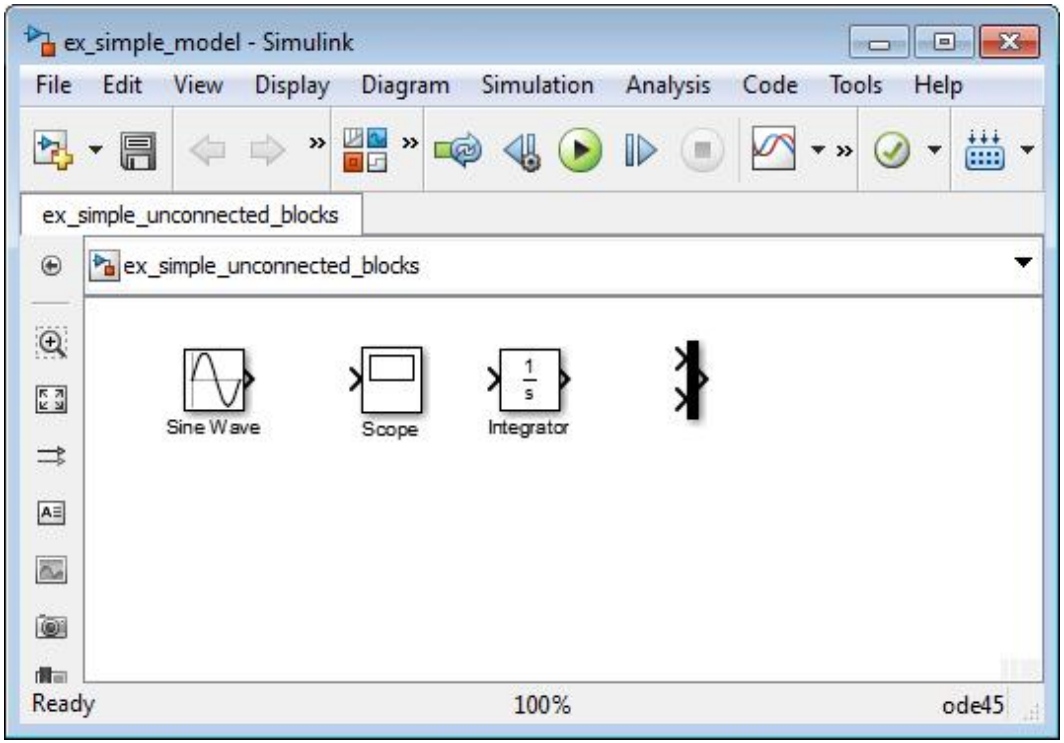


- Add a Scope block using this alternative procedure:
  - Double-click within the block diagram.
  - After the search icon appears, type **scope**, and then from the list, select **Scope**.



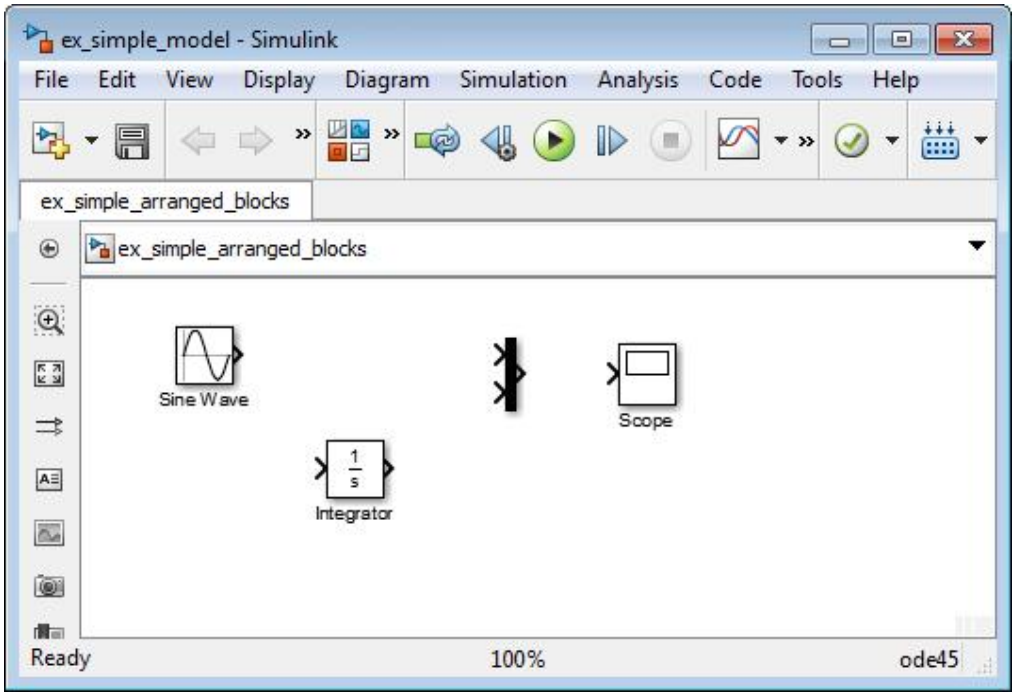
III. c. Add Blocks to a Model

- Add "**Integrator**" and "**Bus Creator**" blocks to your model, using the same approach that you used to add the Sine Wave and Scope blocks.
- Your model should now have the blocks you need for the simple model.



III. d. Move and Resize Blocks

- Before you connect the blocks in your model, arrange them logically to make the signal connections as straightforward as possible.
- Move the Scope block after the Bus block output. You can either:
  - Click and drag a block.
  - Select the block, and then press the arrow keys on your keyboard.
- Move the blocks until your model looks similar to the following figure.



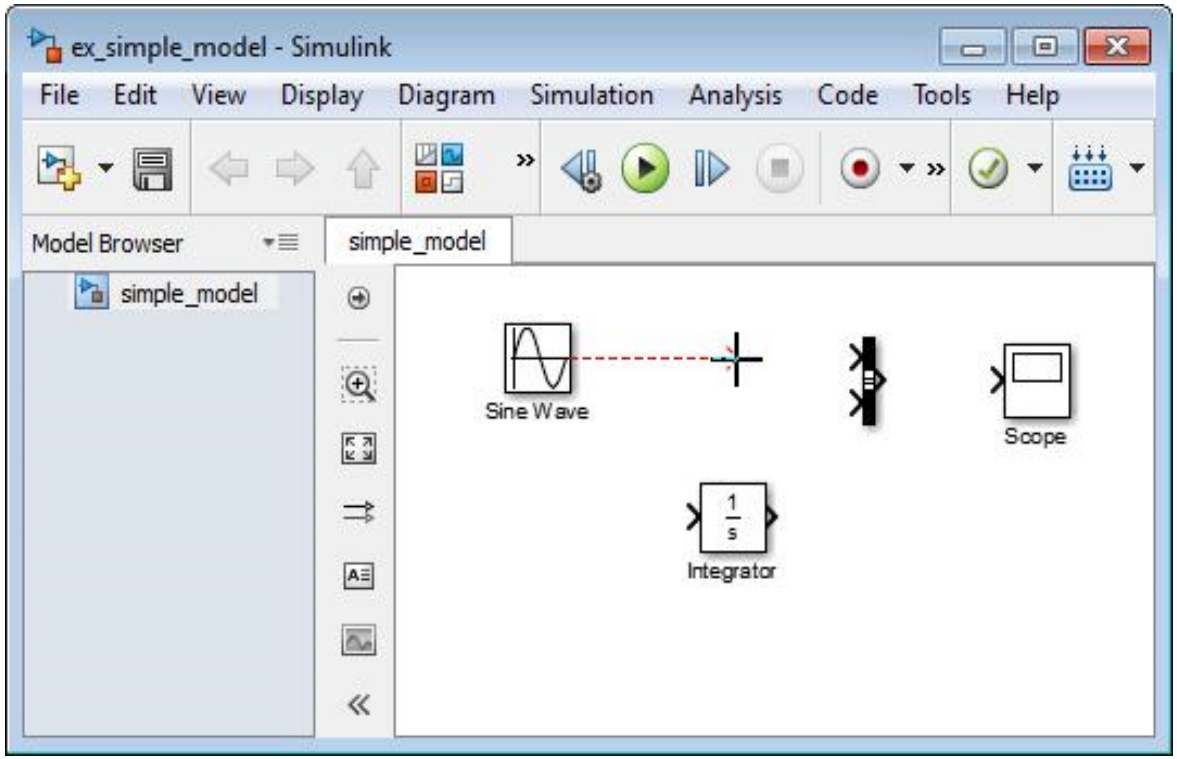
III. e. Simulink Block Connections

- After you add blocks to your model, you can connect them with lines. The connecting lines represent signals within your model.

- Most blocks have angle brackets on one or both sides. These angle brackets represent input and output ports:
  - The **>** symbol pointing into a block is an *input port*.
  - The **>** symbol pointing out of a block is an *output port*.

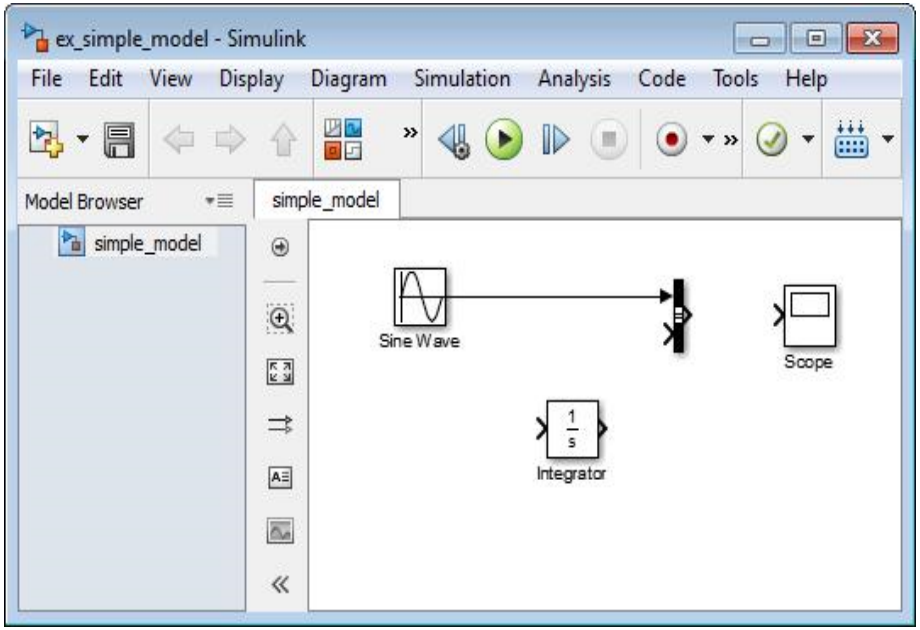
III. f. Draw Signal Lines Between Blocks

- Connect the blocks by drawing lines between output ports and input ports.
- Position the cursor over the output port on the right side of the Sine Wave block. The pointer changes to a cross hair (+) while over the port.
- Click, and then drag a line from the output port to the top input port of the Bus block. While you are holding down the mouse button, the connecting line appears as a red dotted arrow.



III. f. Draw Signal Lines Between Blocks

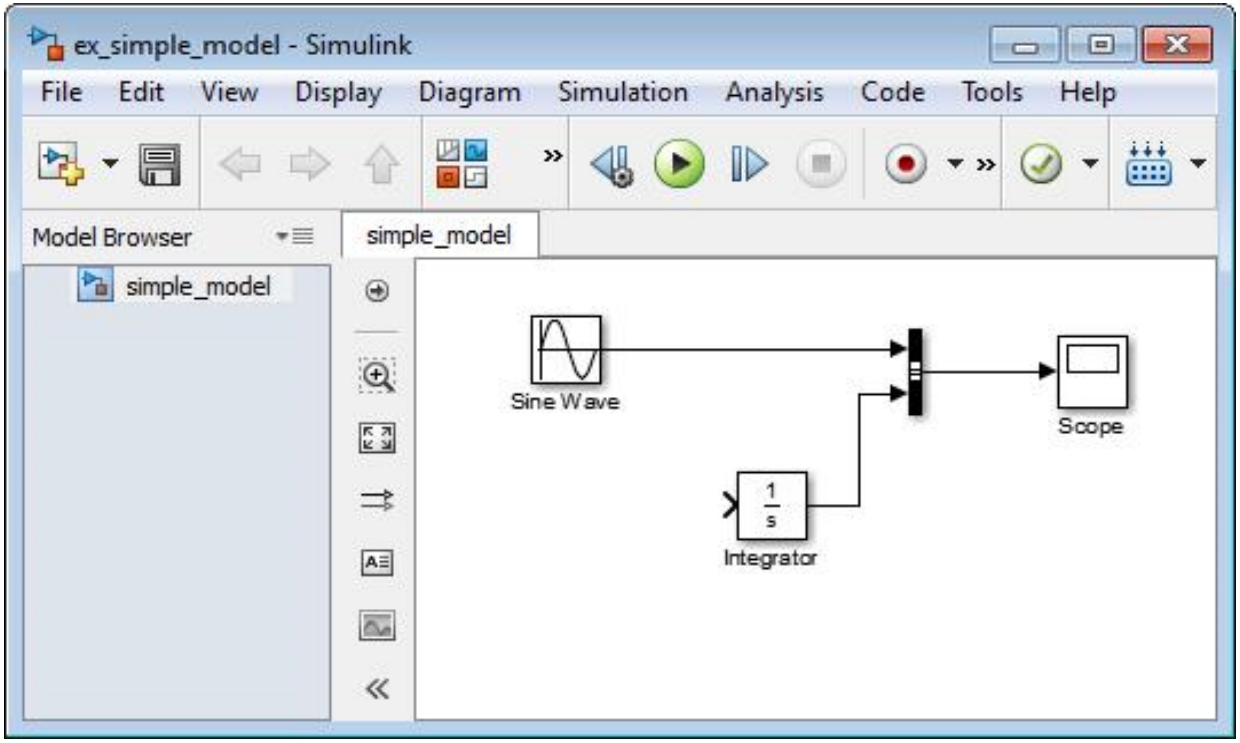
- Release the mouse button when the pointer is over the output port. Simulink connects the blocks with a line and an arrow indicating the direction of signal flow.



- Connect the output port of the Integrator block to the bottom input port on the Bus Connector block using this alternative procedure:
  - Select the Integrator block,
  - Press and hold the **Ctrl** key,
  - Click the Bus Connector block.

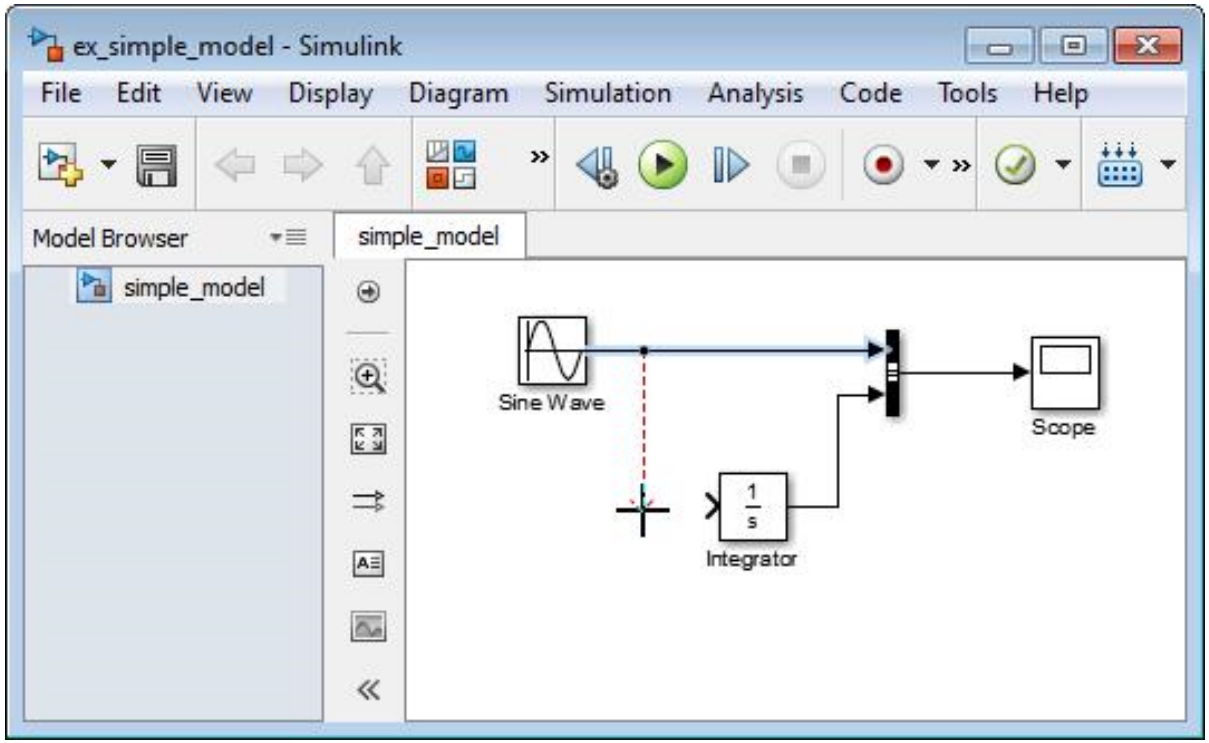
III. f. Draw Signal Lines Between Blocks

- The Integrator block connects to the Bus Connector block with a signal line
- Connect the Bus Connector block to the Scope block by aligning ports:
  - Click and drag the Scope block until its input port is aligned with the Bus Connector output port. A light blue line appears between the ports.
  - Release the mouse button, and then click the blue arrow. The blue arrow changes to a black signal line connecting the blocks.



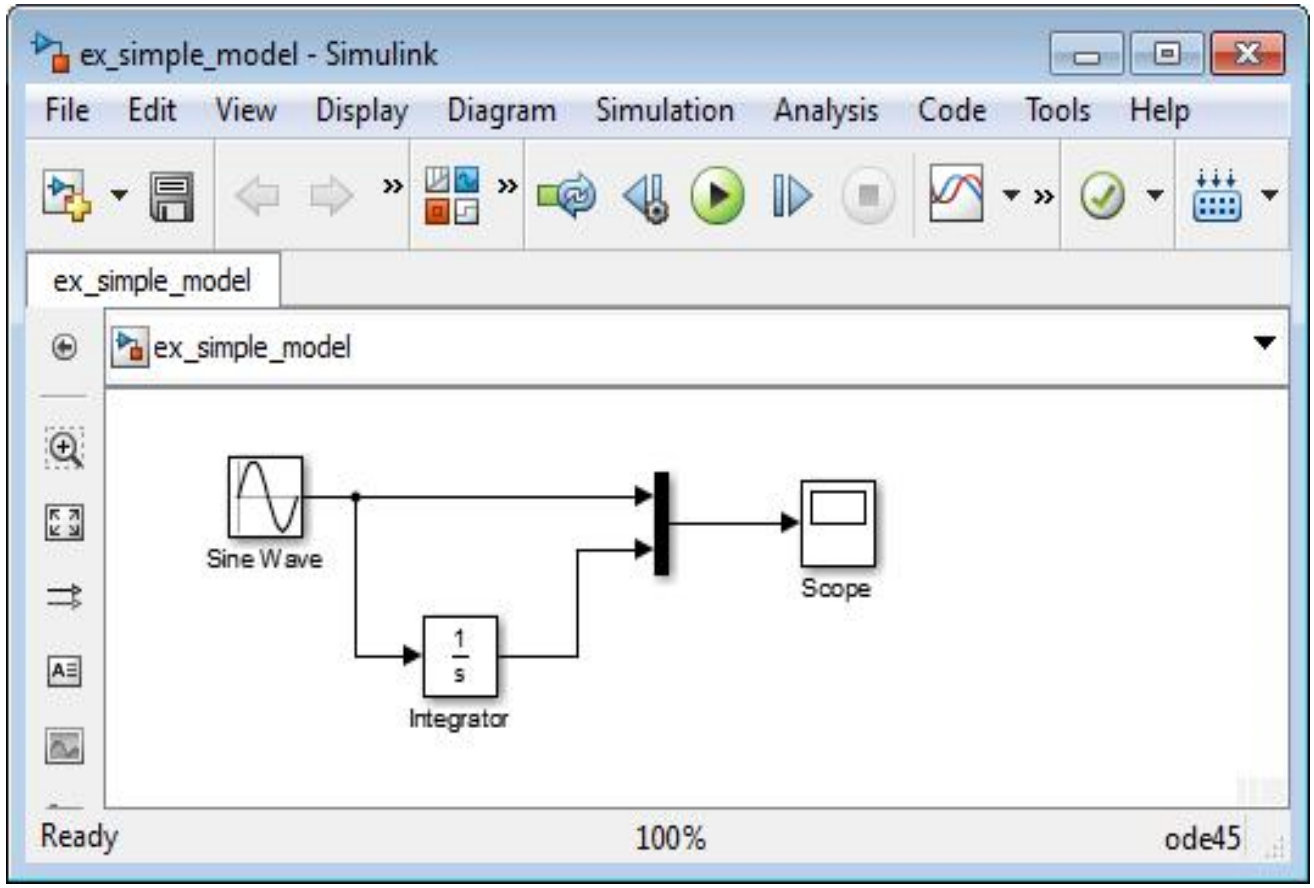
III. g. Draw Branched Signal Lines

- Your simple model is almost complete. To finish the model, connect the Sine Wave block to the Integrator block. This connection is different from the other connections, which all connect output ports to input ports.
- Hold down the **Ctrl** key.
- Position the cursor where you want to start a branch line. Click, and then drag the cursor away from the line to form a dotted-red line segment.



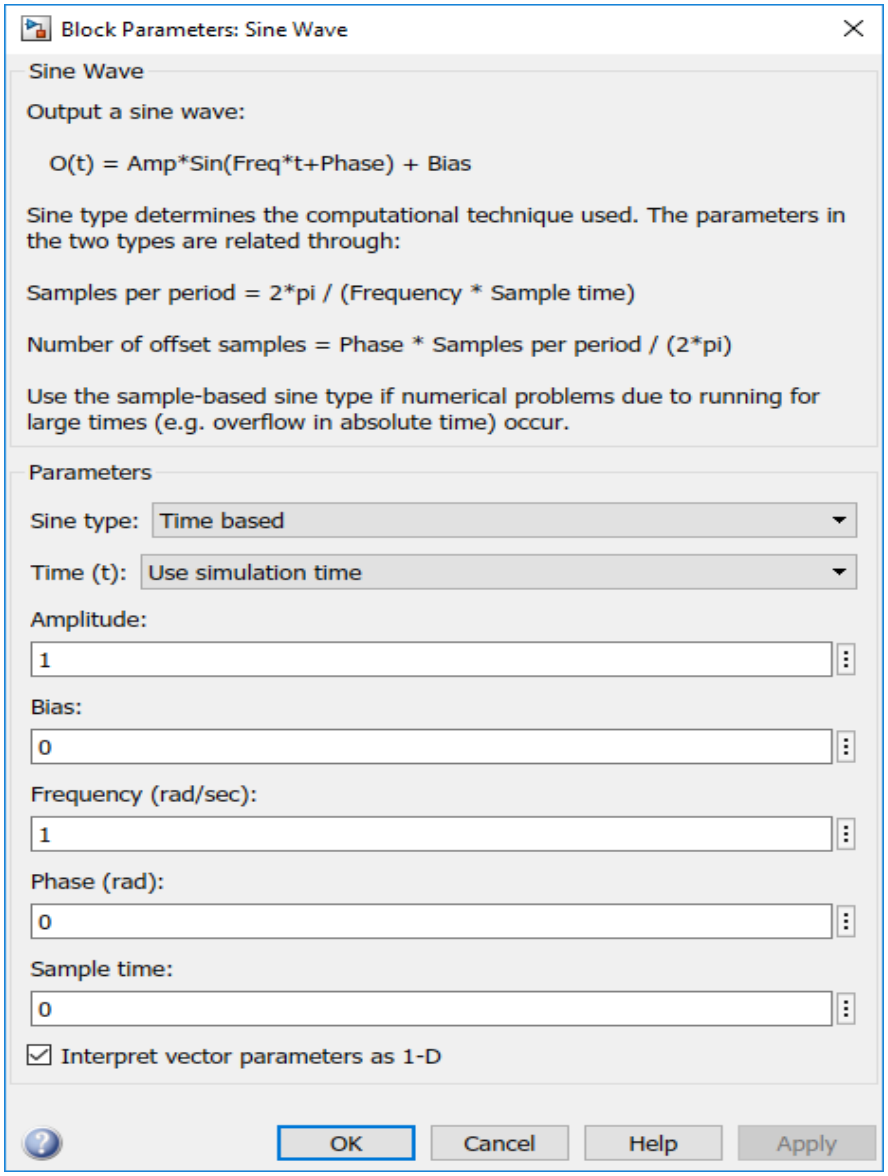
III. g. Draw Branched Signal Lines

- Drag the cursor to the Integrator input port, and then release the mouse button. The new line, called a branch line, carries the same signal that passes from the Sine Wave block to the Bus block.
- Drag line segments to straighten and align with blocks. Your model is now complete.



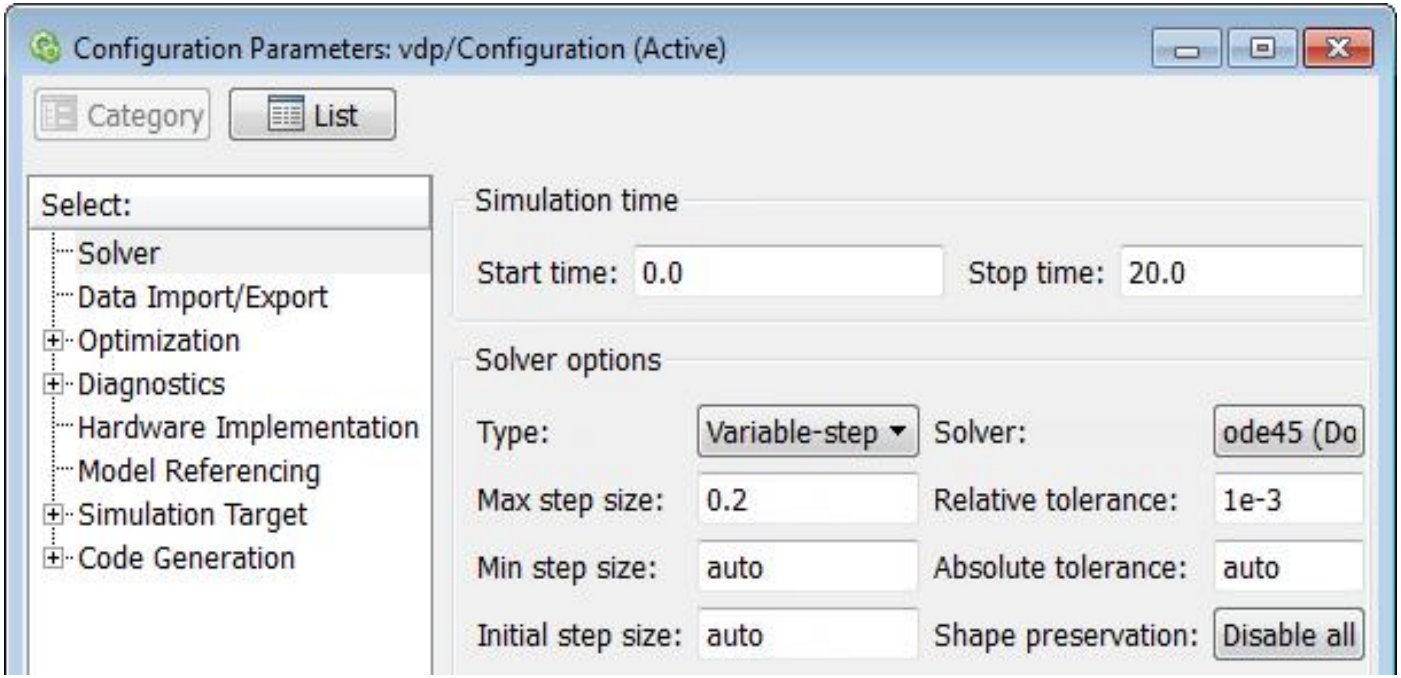
III. h. Define Simulation and Block Parameters

- Before you simulate the behavior of a model, define the simulation parameters. Simulation parameters include the type of numerical solver, start, and stop times, and maximum step size.
- Double-click Sine Wave block to open Block Parameters window. The parameters of sine wave (e.g., amplitude, frequency, phase, etc.) can be changed. For the simple model, the default parameters are used for the simulation



III. h. Define Simulation and Block Parameters

- From the Simulink Editor menu, select **Simulation > Model Configuration Parameters**. The Configuration Parameters dialog box opens to the Solver pane.
- In the Stop time field, enter **20**. In the **Max step size** field, enter **0.2**.



- Click **OK**.

III. i. Run Simulation

- After you define the Model Configuration Parameters, you are ready to simulate your model.

- From the Simulink Editor menu bar, select **Simulation > Run**. The simulation runs, and then stops when it reaches the stop time specified in the Model Configuration Parameters dialog box.

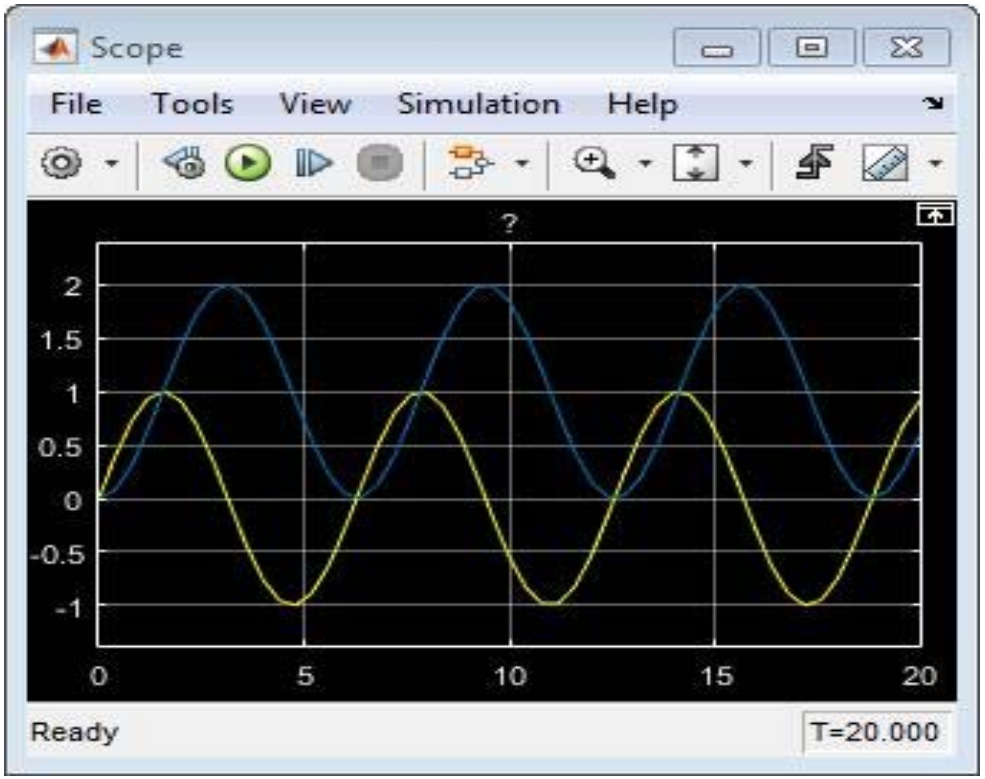
- Alternatively, you can control a simulation by clicking
  - **Run** simulation button 

  and
  - **Pause** simulation button 

  on the Simulink Editor toolbar.

III. j. Observe Simulation Results

- After simulating the simple model, you can view the simulation results in a Scope window.
- Double-click the Scope block.
- The Scope window opens and displays the simulation results. The plot shows a sine wave signal with the resulting integrated sine wave signal.



Tutorial 2:

Wireshark Tutorial

Wireshark: Introduction

- Wireshark is a free network packet analyzer which captures network packets and displays the contents of all fields within a packet.
- In fact, this tool allows you to observe the sequence of messages exchanged between two protocol entities which helps to obtain a deeper understanding of network protocol operations.
- This document provides information required to get Wireshark started. The contents are taken from the "Wireshark User's Guide" available at https://www.wireshark.org/docs/wsug_html/.

Getting Wireshark

- **Downloading Wireshark**

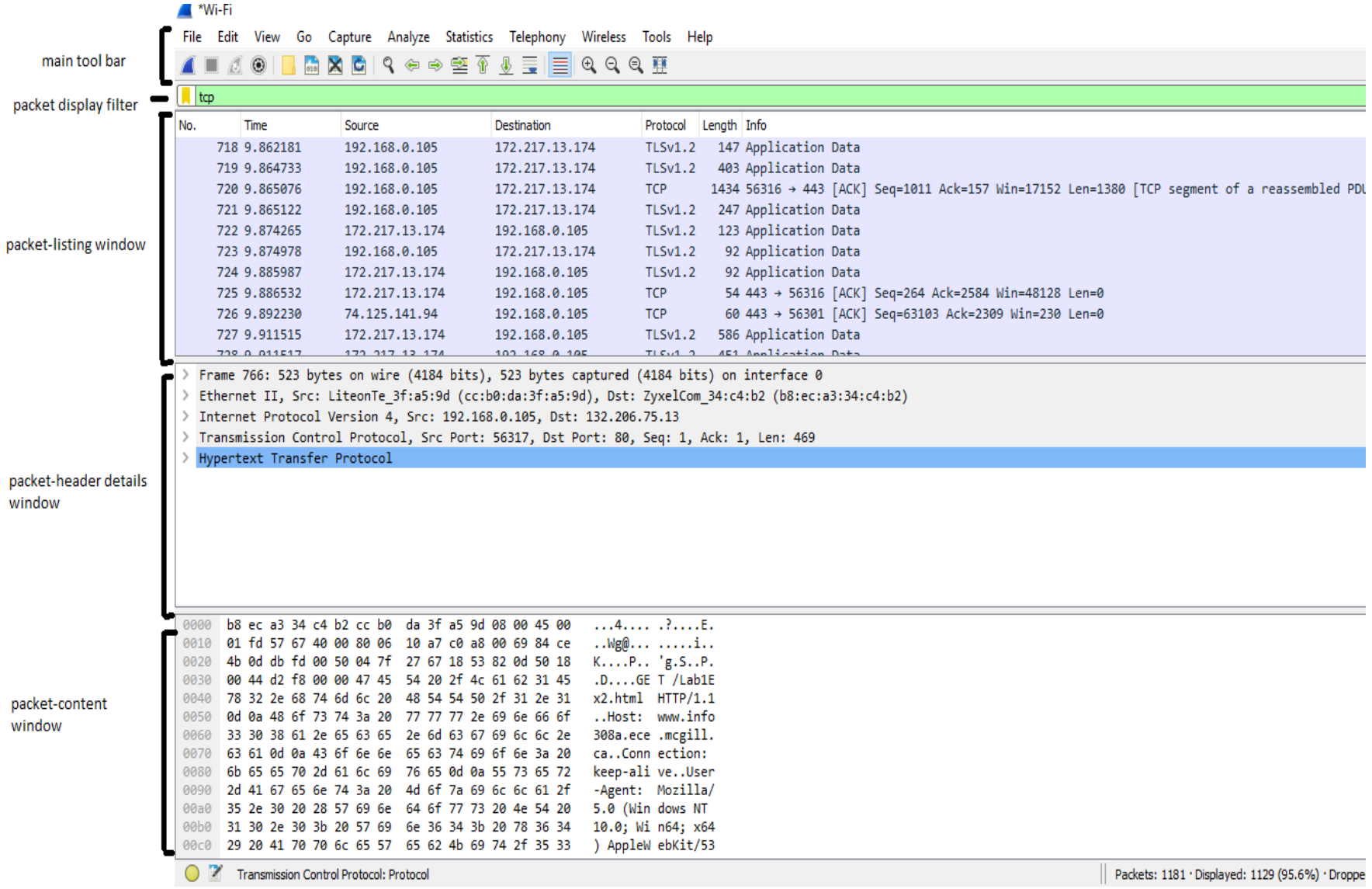
  You can get the latest copy of the program from the Wireshark website at:

  https://www.wireshark.org/download.html

  If you have troubles installing or running Wireshark, you can visit Wireshark FAQ https://www.wireshark.org/faq.html which provides a number of helpful hints and interesting tidbits of information.

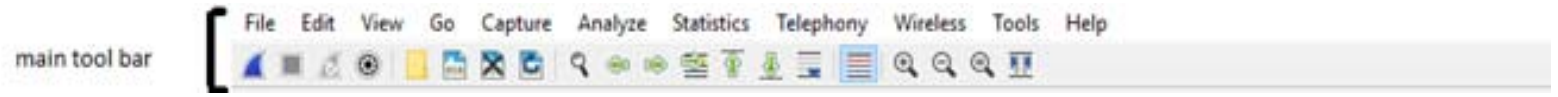
Wireshark user interface: overview

The Wireshark interface has five major components:

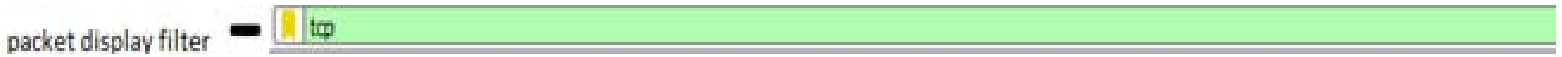


Main toolbar & Packet display filter

1. **Main toolbar:** The main toolbar provides quick access to frequently used items from the menu. This toolbar cannot be customized by the user, but it can be hidden using the View menu, if the space on the screen is needed to show even more packet data. Only the items useful in the current program state will be available. The others will be greyed out (e.g. you cannot save a capture file if you haven't loaded one).



2. **Packet display filter:** You can enter a protocol name or other information into this field to filter the information displayed in the packet-listing window.

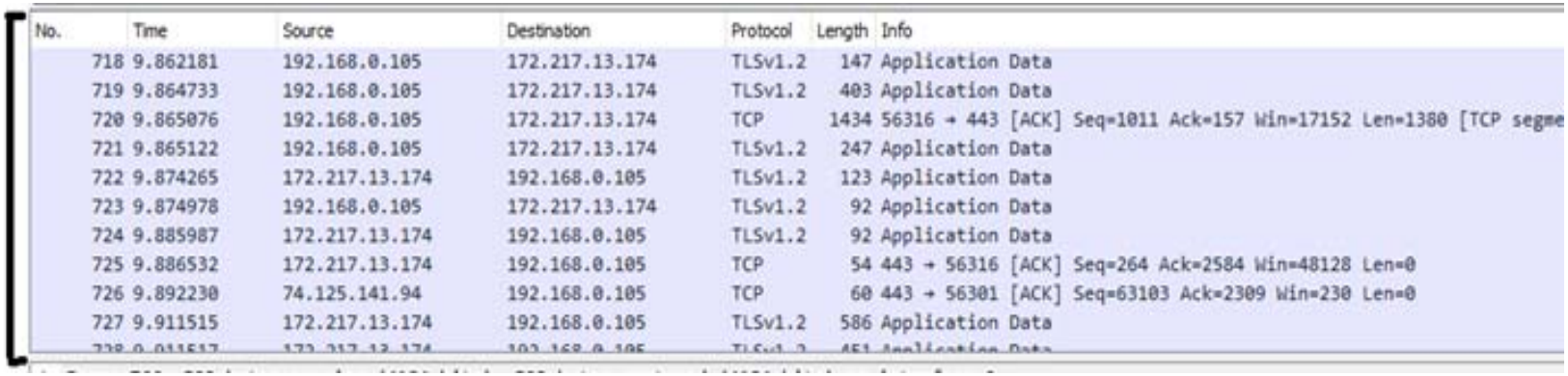


Packet-listing window

3. **Packet-listing window:**

- The packet-listing window displays all the packets in the current capture file. Each line in the packet list corresponds to one packet in the capture file. If you select a line in this window, more details will be displayed in the "Packet-header Details" and "Packet Content/Packet Byte" windows.
- While dissecting a packet, Wireshark will place information from the protocol dissectors into the columns. As higher level protocols might overwrite information from lower levels, you will typically see the information from the highest possible level only.

packet-listing window



No.	Time	Source	Destination	Protocol	Length	Info
718	9.862181	192.168.0.105	172.217.13.174	TLSv1.2	147	Application Data
719	9.864733	192.168.0.105	172.217.13.174	TLSv1.2	403	Application Data
720	9.865076	192.168.0.105	172.217.13.174	TCP	1434	56316 → 443 [ACK] Seq=1011 Ack=157 Win=17152 Len=1380 [TCP segme
721	9.865122	192.168.0.105	172.217.13.174	TLSv1.2	247	Application Data
722	9.874265	172.217.13.174	192.168.0.105	TLSv1.2	123	Application Data
723	9.874978	192.168.0.105	172.217.13.174	TLSv1.2	92	Application Data
724	9.885987	172.217.13.174	192.168.0.105	TLSv1.2	92	Application Data
725	9.886532	172.217.13.174	192.168.0.105	TCP	54	443 → 56316 [ACK] Seq=264 Ack=2584 Win=48128 Len=0
726	9.892230	74.125.141.94	192.168.0.105	TCP	60	443 → 56301 [ACK] Seq=63103 Ack=2309 Win=230 Len=0
727	9.911515	172.217.13.174	192.168.0.105	TLSv1.2	586	Application Data

default columns

There are a lot of different columns available. The default columns will show:

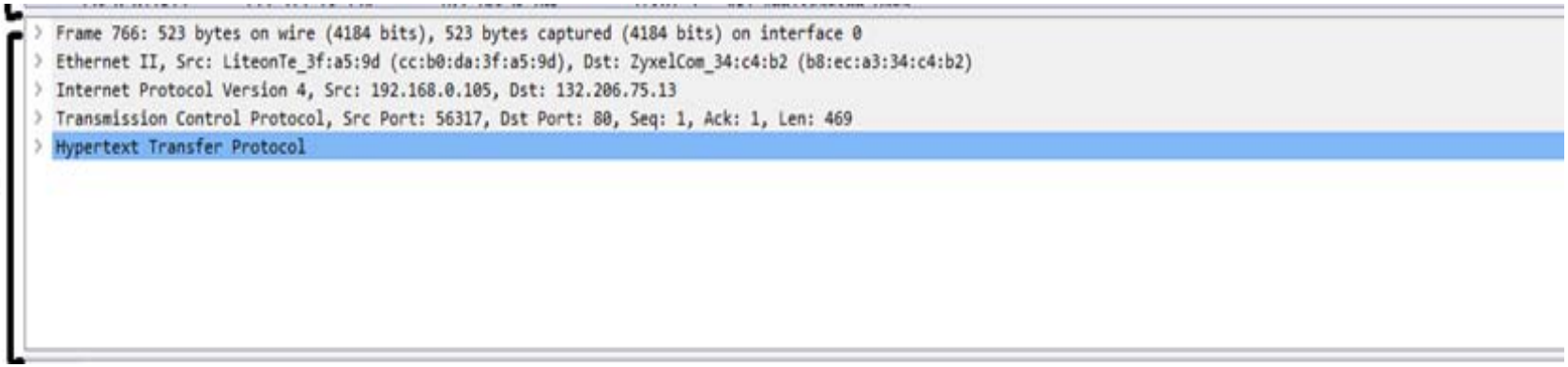
- **No.** The number of the packet in the capture file. This number won't change, even if a display filter is used.
- **Time** The timestamp of the packet.
- **Source** The address where this packet is coming from.
- **Destination** The address where this packet is going to.
- **Protocol** The protocol name in a short (perhaps abbreviated) version.
- **Length** The length of each packet.
- **Info** Additional information about the packet content.

The first column shows how each packet is related to the selected packet.

Packet-header details window

4. **Packet-header details window:**

   The packet-header details window shows the current packet (selected in the packet-listing window) in a more detailed form. This window shows the protocols and protocol fields of the packet selected in the packet-listing window. The protocols and fields of the packet shown in a tree which can be expanded and collapsed.



Generated fields & Links

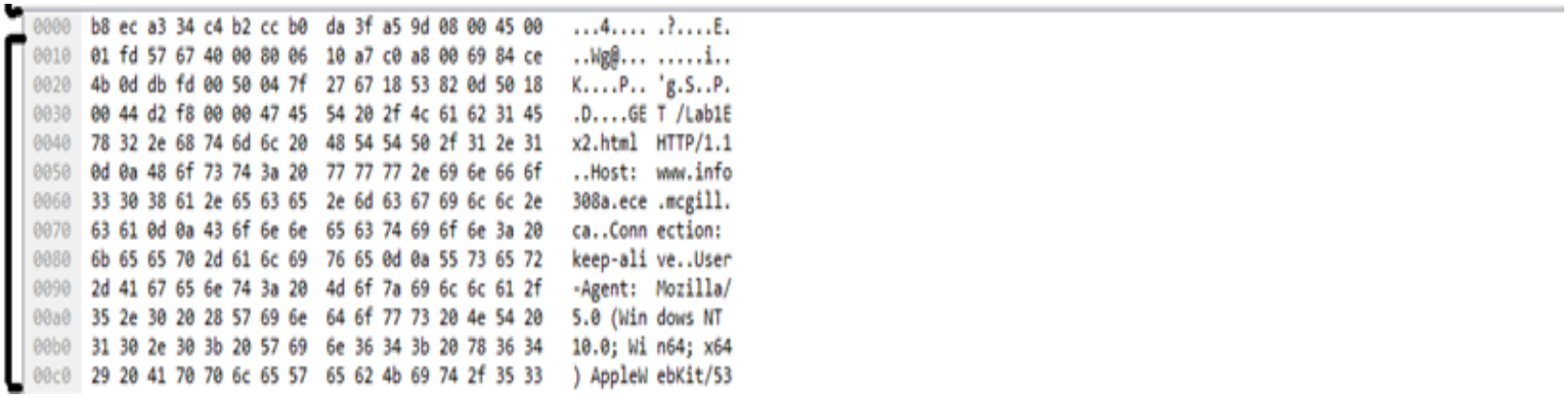
Some protocol fields have special meanings.

- **Generated fields** Wireshark itself will generate additional protocol information which isn't present in the captured data. This information is enclosed in square brackets ('[' and ']'). Generated information includes response times, TCP analysis, GeoIP information, and checksum validation.
- **Links.** If Wireshark detects a relationship to another packet in the capture file it will generate a link to that packet. Links are underlined and displayed in blue. If you double-clicked on a link Wireshark will jump to the corresponding packet.

Packet-content window

5. **Packet-content window:** The packet-content window shows the data of the current packet (selected in the packet-listing window) in a hexdump style. Each line contains the data offset, sixteen hexadecimal bytes, and sixteen ASCII bytes. Non-printable bytes are replaced with a period ('.').

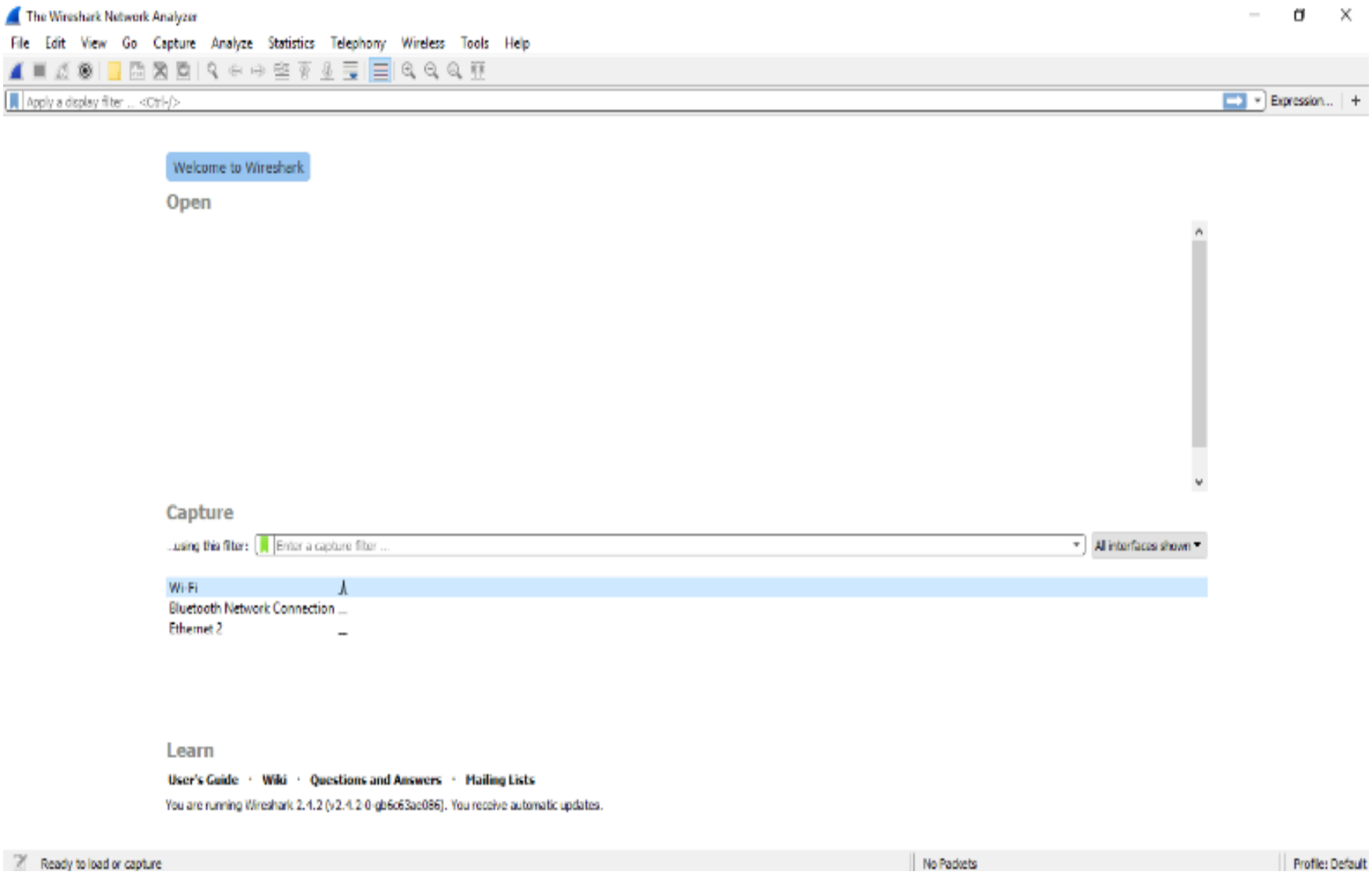
packet-content window



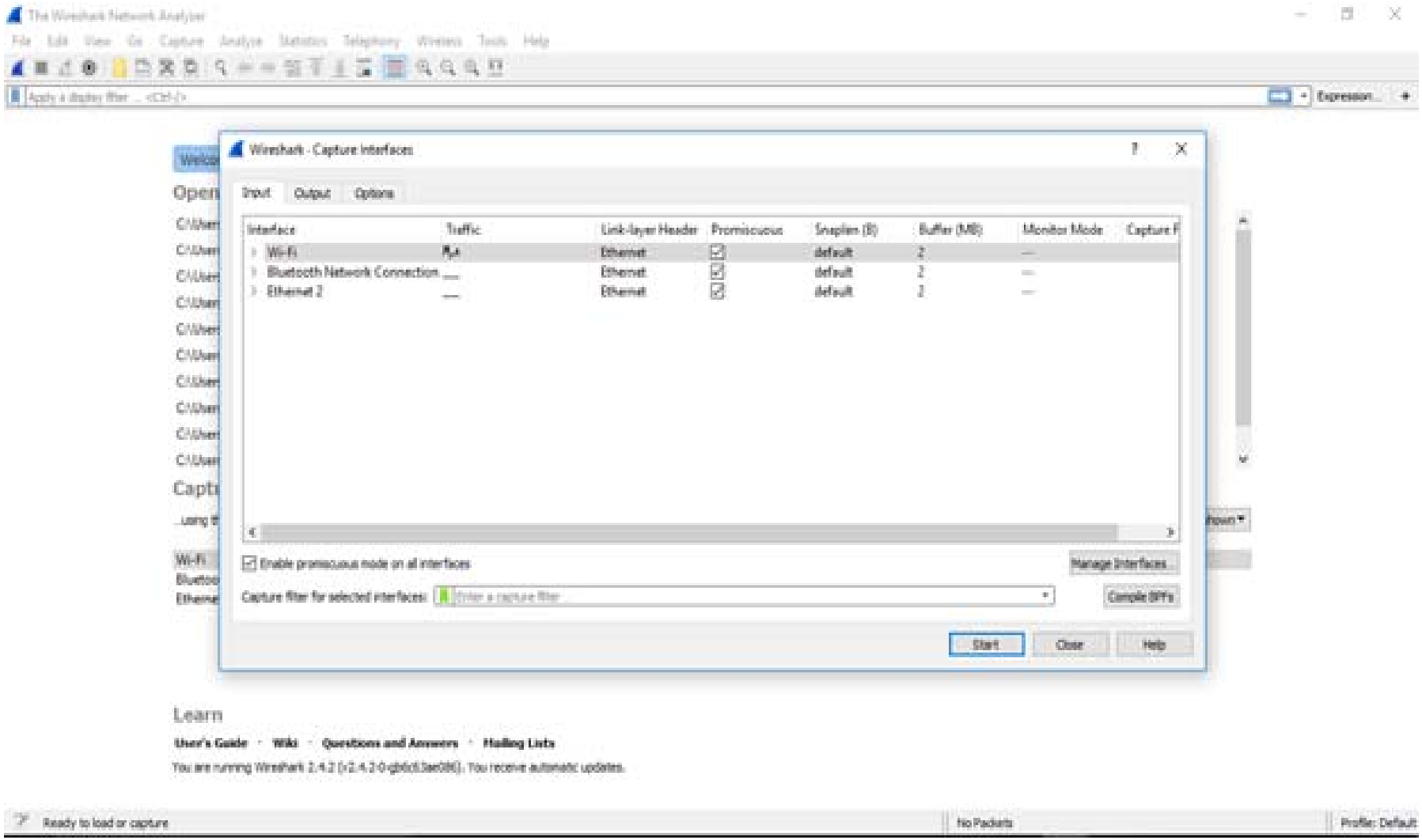
Starting packet capturing (1/3)

- You can double-click on an interface in the main window.
- You can get an overview of the available interfaces using the “Capture Interfaces” dialog box (**Capture** → **Options…**). You can start a capture from this dialog box using the **Start** button.
- You can immediately start a capture using your current settings by selecting **Capture** → **Start** or by clicking the first toolbar button.

Starting packet capturing (2/3)



Starting packet capturing (3/3)



Test (1/2)

- Start up your web browser.
- Open up the Wireshark. In the filter field, type “http” (without the quotation marks), so that only captured HTTP messages will be displayed later in the packet-listing window.

  Note: After you have changed the expression in the filter input box, do not forget to press the Apply button (or the Enter/Return key twice), to apply this filter string to the displayed trace file.
- Start Wireshark packet capture.
- Go back to your web browser and enter:

  http://www.cbc.ca/news
- Stop Wireshark packet capture.

Test (2/2)

You should see a Wireshark window that looks like this: