Microwave Engineering Practices

Lab Instructor: Dr. Wu Guang

Lab Timing: Thursday (10:00 ─ 12:00)

Experiment: 2

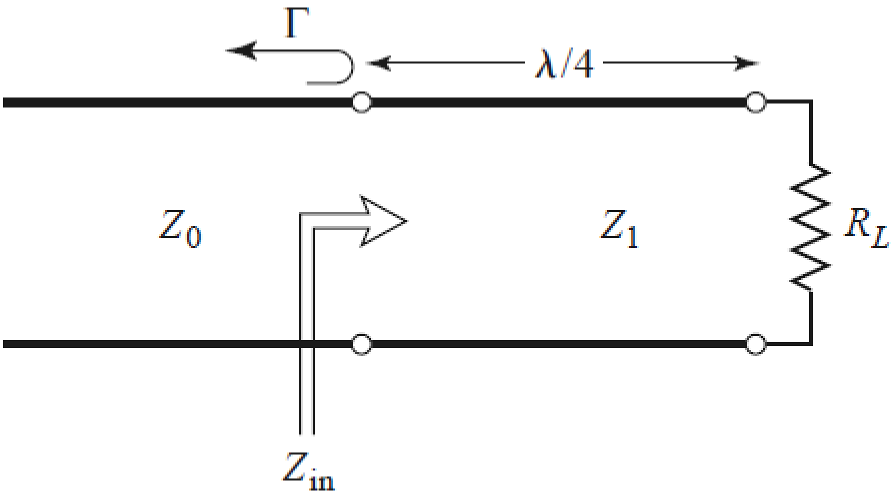
Introduction to Advanced Design System: Design of Impedance Transformers

Southern University of Science and Technology, Shenzhen, P.R. China

Task-1:

Design of Quarter-wavelength Impedance Transformer

Design an impedance transformer to transform a load impedance of $\boldsymbol{R_L}$ **= 100 Ω** to a source impedance of $\mathbf{Z_S} = \mathbf{Z_0}$ **= 50 Ω** at the center frequency of **1.0 GHz**. Microstrip technology can be used to develop the impedance transformer and Advanced Design System 15.01 software can be used to perform the simulation.



Calculation:

Electrical length = λ/4 or 90° $\left\{\text{as you know TL length} = \beta L = \frac{2\pi}{\lambda} L = \frac{2\pi}{\lambda} \times \frac{\lambda}{4} = 90^{\circ}\right\}$

$R_L = 100\ \Omega$, $Z_0 = 50\ \Omega$, Z_1 =?

The electrical length of the transmission line is λ/4 (also called quarter-wavelength), the characteristic impedance can be calculated as:

$$Z_1 = \sqrt{R_L Z_0} = \sqrt{100 \times 50} = 70.71\,\Omega$$

Task-2:

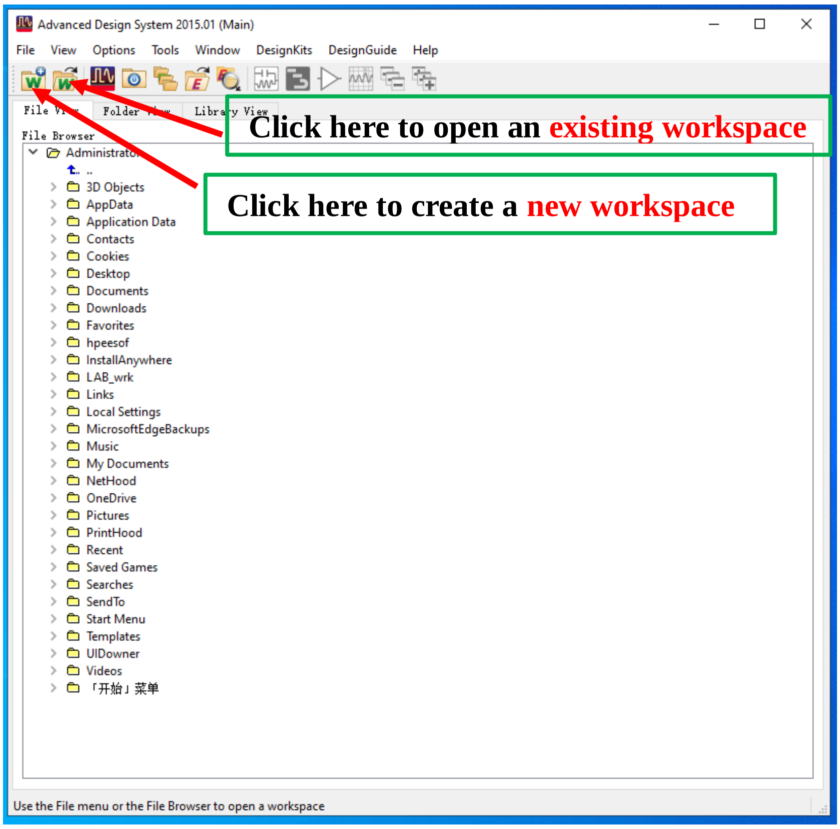
Design of L-Section Impedance Transformer

Design a L-section impedance transformer to transform a load impedance of $\boldsymbol{Z_L}$ **= 60+j80 Ω** to a source impedance of $\mathbf{Z_S} = \mathbf{Z_0}$ **= 50 Ω** at the center frequency of **2.0 GHz**. Microstrip technology can be used to develop the impedance transformer and Advanced Design System 15.01 software can be used to perform the simulation.

ADS Design Environment

Launching Advanced Design System (ADS)

To access Agilent Advanced Design System (ADS), double click on the ADS icon from the desktop.

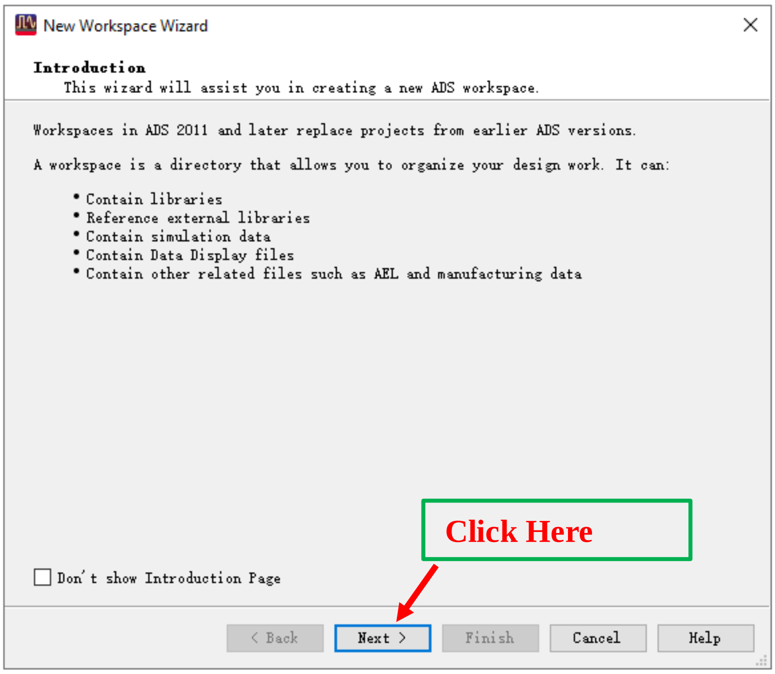


ADS Main Window

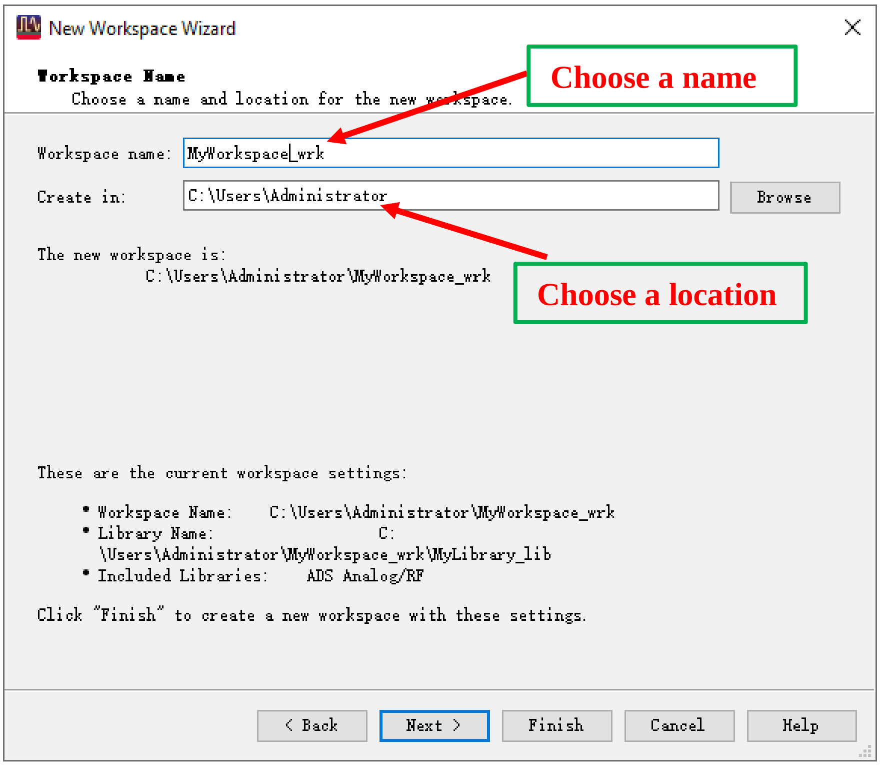
The ADS Main Window enables you to create and manage ADS workspaces. A workspace is an organizer where you can group everything about a design within— such as libraries, technology, schematic, layout, simulation data, and Momentum data. The entire ADS user interface and simulation operates within a currently opened workspace. The File View page of the Main window allows you to traverse to your existing workspaces (located in the default folder). To open any of these workspaces, select and right-click on the workspace and choose Open Workspace or double-click on the selected **workspace**.

Create New Workspace:

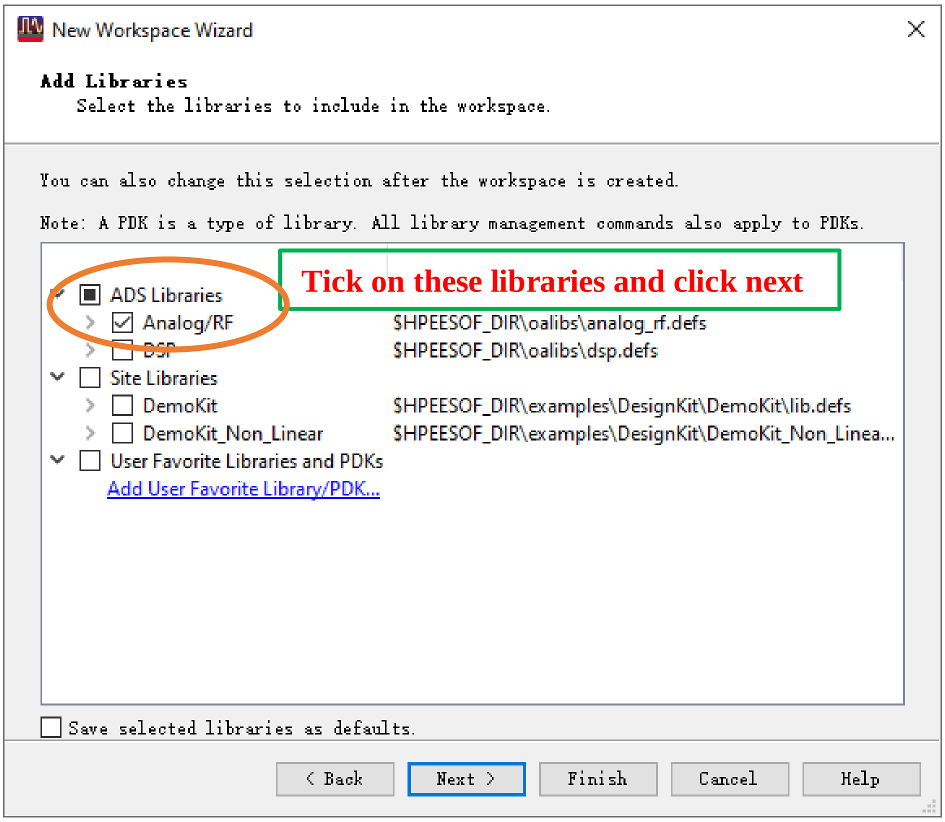
1. In an ADS Main Window, **click** the  on standard tool bar **or** select the menu item **File > New > Workspace**
2. **New Workspace Wizard** will appear, **click next** tab present at the bottom of the New Workspace Wizard window



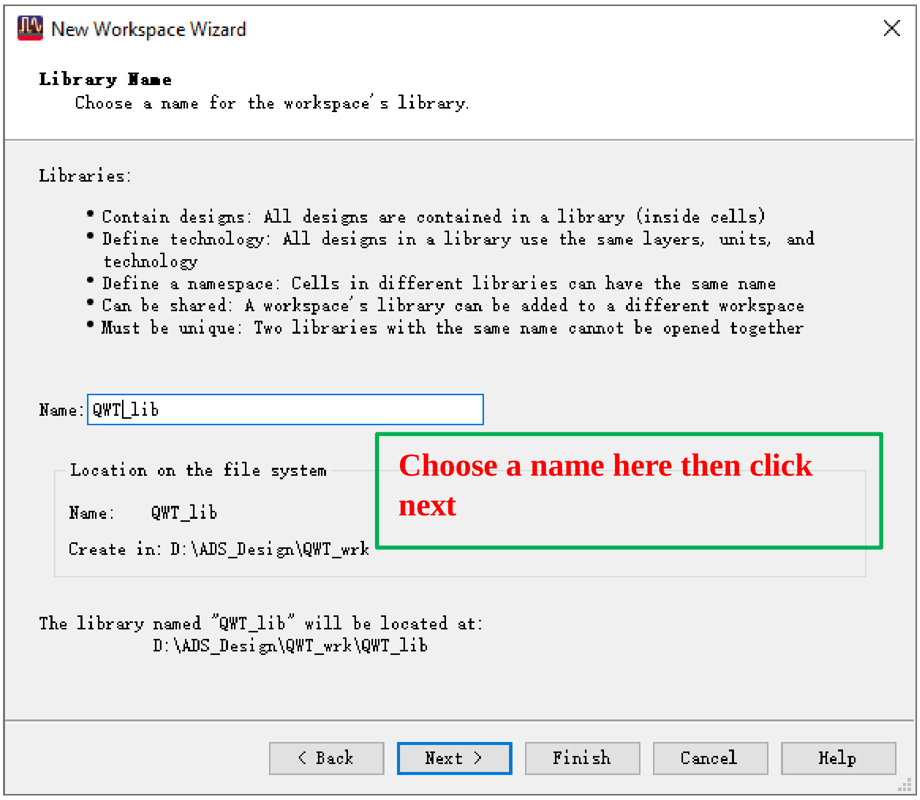
3. Choose a name and location for the new workspace



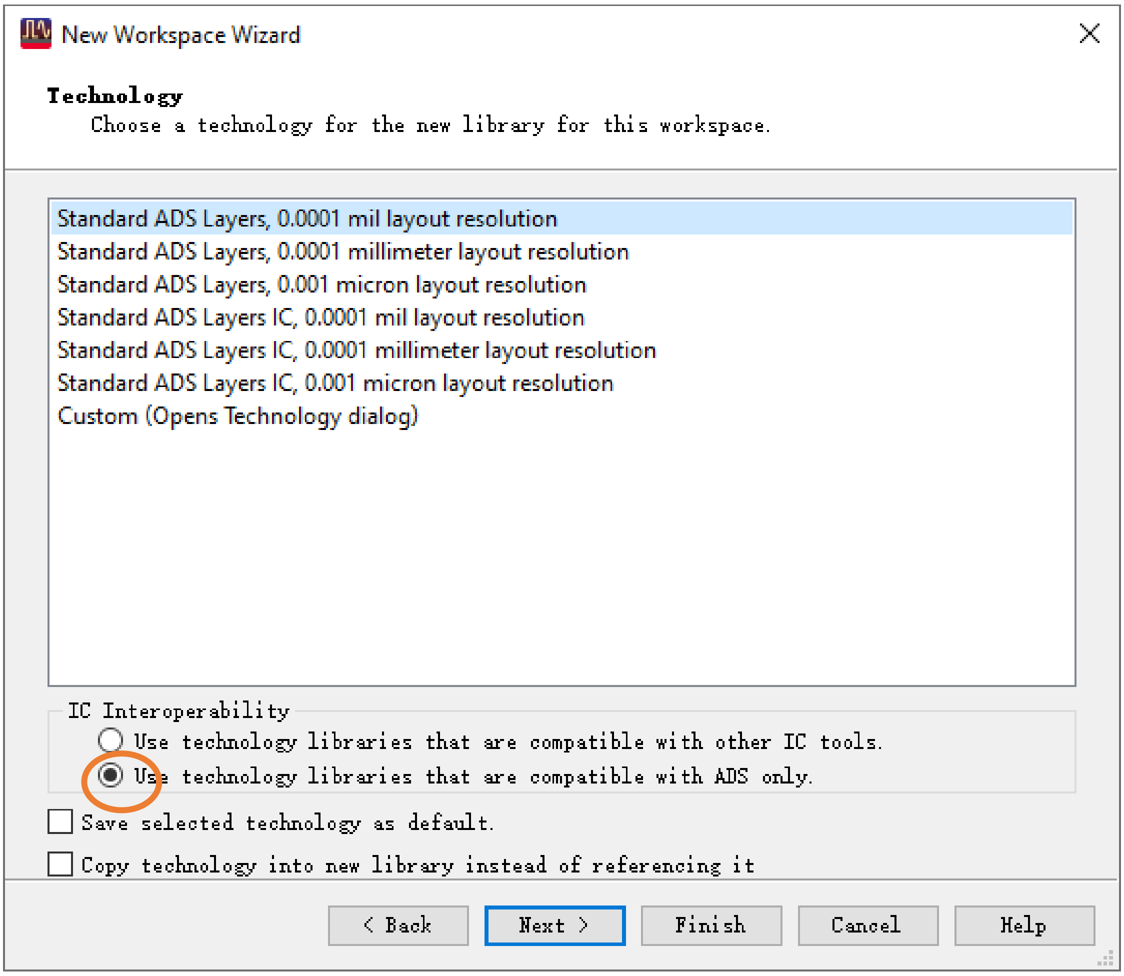
4. **Add Libraries**: select the libraries to include in the workspace.



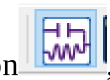
5. **Library name**: choose a name for the workspace's library

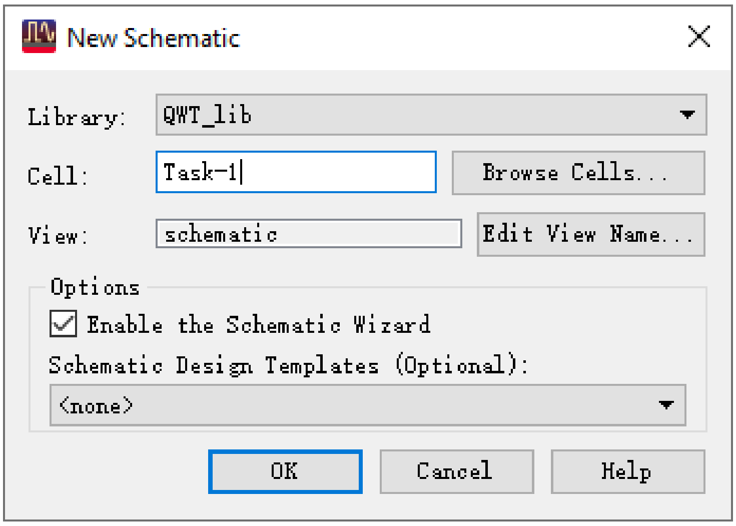


6. **Select a Technology:** choose default technology and **click next** then **click Finish**.

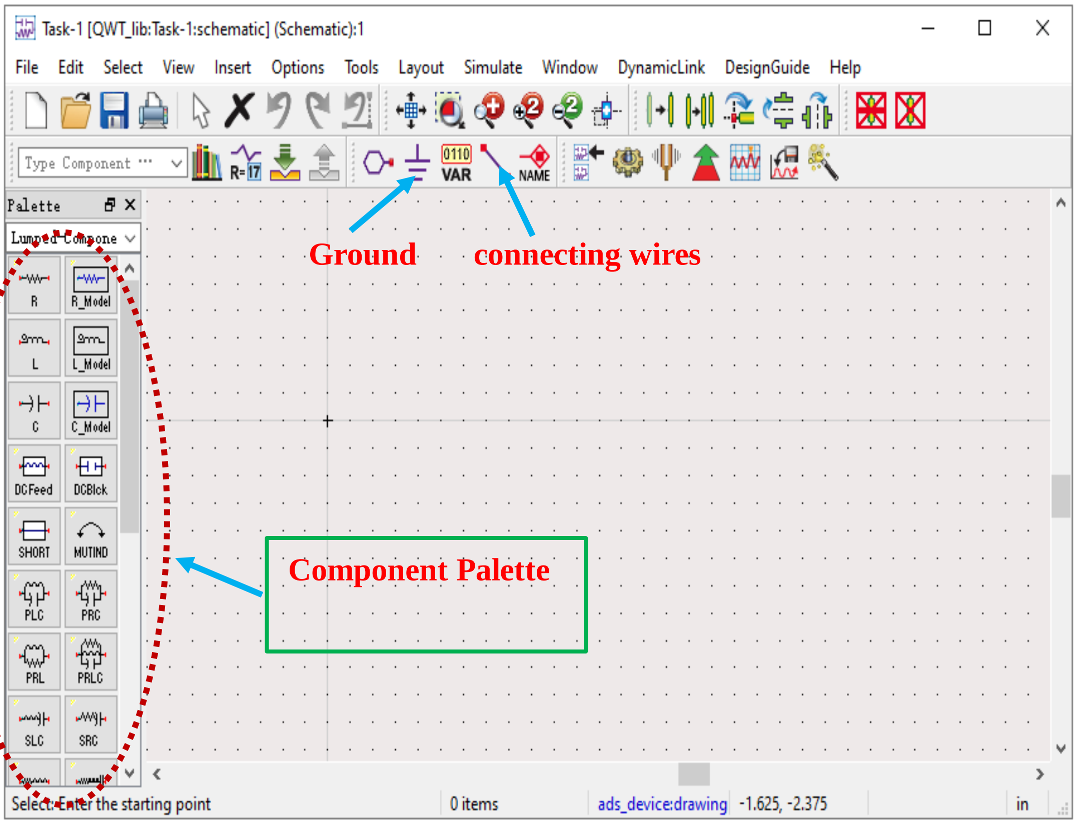


Creating a Schematic Design

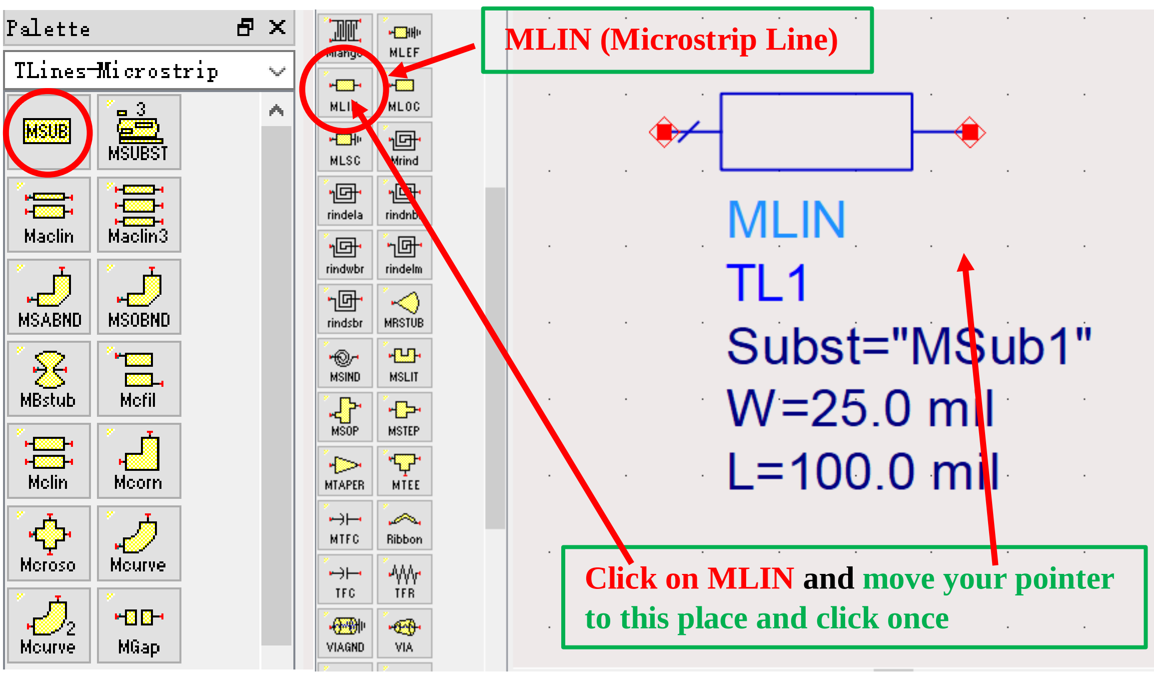
1. In the Main window, click the **New Schematic Window** icon , or select the menu item ***Window*** > ***New Schematic Window*** or **ctrl+shift+N**. Immediately, the Schematic window will appear.



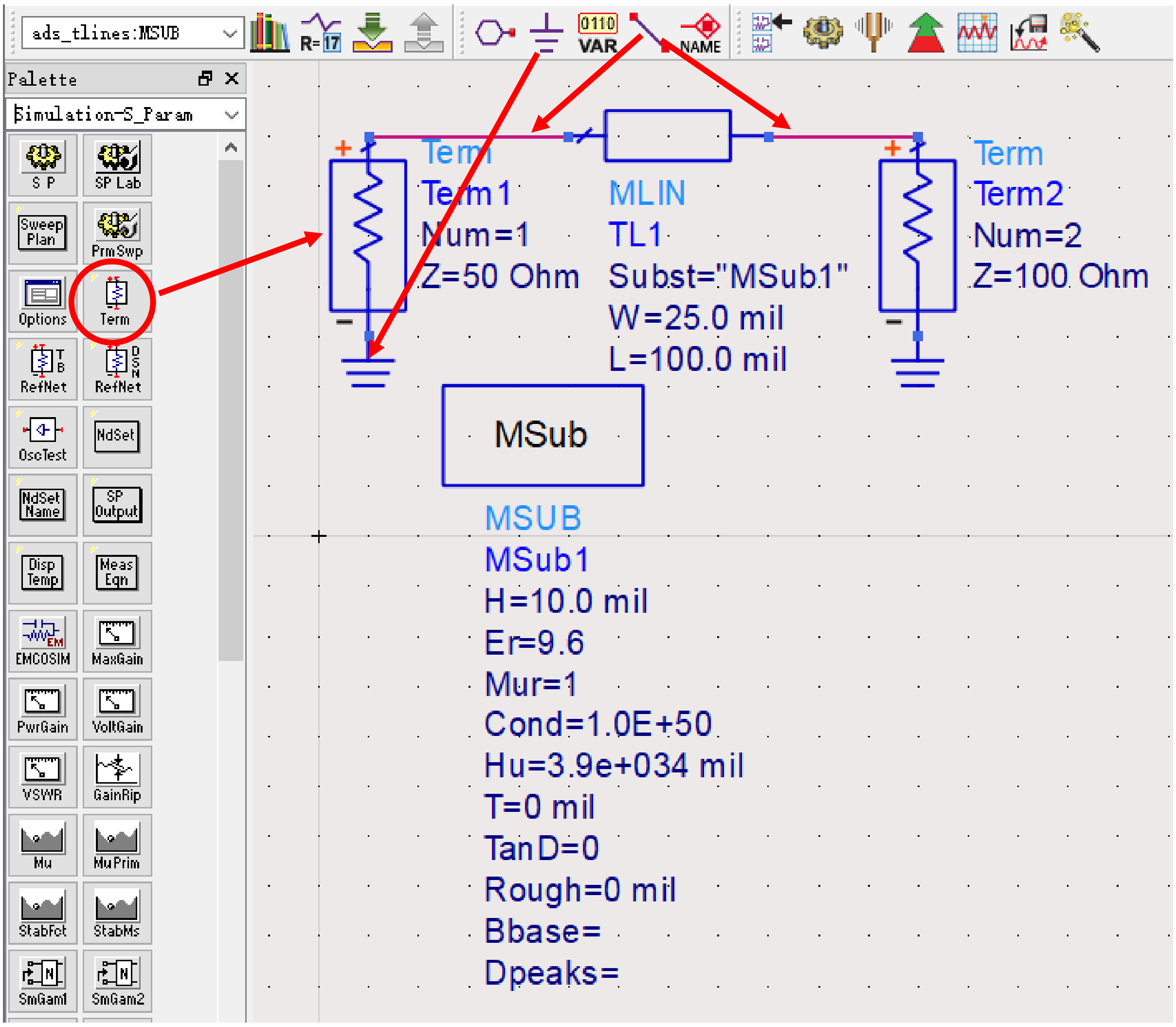
2. Choose a **Name** for your project and save the **schematic design** by click on **OK**.



3. In the Components Palette List, click the **TLines-Mcrostrip** palette.
4. In the **TLines-Microstip** palette, click on the **MLIN**. Then click to insert the **MLIN** as shown on the schematic. Next, insert the other five MLINs.



5. In the **TLines-Microstip** palette, click on the **MSUB** (**Microtrip Substrate**). Then click to insert the **MSUB** on the schematic.
6. Select the **Simulation-S_Param** palette to insert the terminations (**TERM**). Then insert two terminations.
7. Insert **grounds** and **connecting wires**. The **ground** and **wire icon** are placed in the top toolbar as shown above.
8. Combine all the components by using **wire** as shown below.

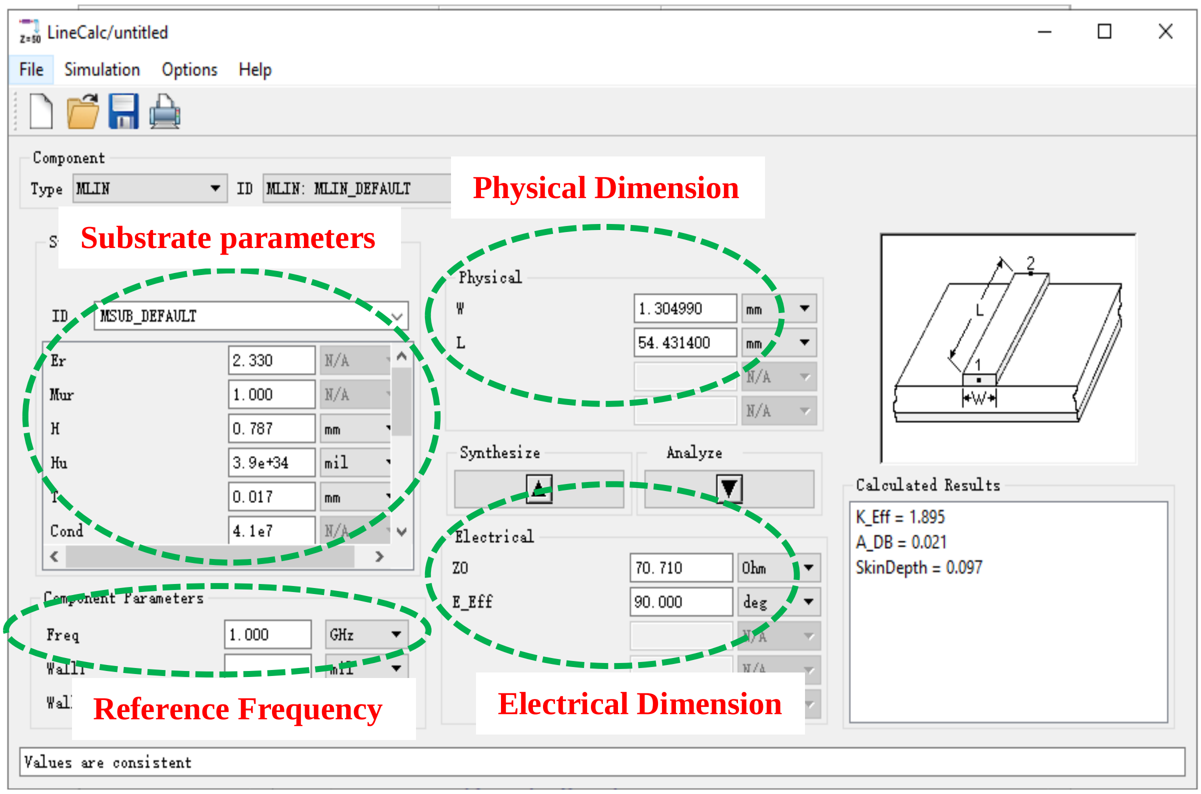


9. After the circuit is built, edit the parameters of the microstrip substrate (**MSUB**). To do this, double-click on the **MSUB**. Then edit the substrate parameters as shown in the table below.

Substrate			Conductor	
Dielectric constant	Substrate thickness	Loss tangent	Copper thickness	Conductivity
2.33	0.787 mm	0.0012	0.017 mm	5.8×10^{7}

*Use **LineCalc** to determine the physical parameters of the MLINs*

1. From the schematic window start **LineCalc**: **Tools** > **LineCalc** > **Start LineCalc**. The main window will appear as shown here.



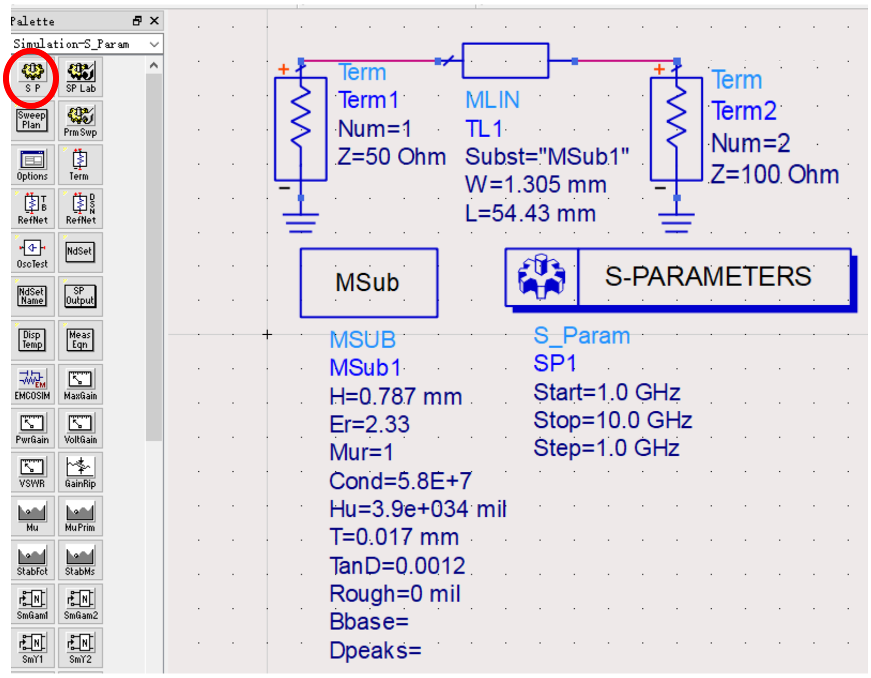
NOTE on LineCalc: Notice that the default component type is MLIN and that the default substrate is an ADS MSUB but it may not be set to the values on your schematic.

2. **Substrate Parameters**: Set the **MSUB** values to your schematic MSUB values, such as H=0.787 mm, etc.
3. Component Parameters: Set **Freq**. to 1 GHz (this will be easier to use with the reference line and the numbers will be easier to use).
4. Electrical: Set the Electrical characteristic (characteristic impedance and electrical length) of a microstrip line. Here, set Z0 = 70.71 ohm and E_Eff =90°.
5. Click the **Synthesize button** and the width and length of the microstrip line will be calculated.

Components	Characteristic impedance	Electrical length	Microstrip Width	Microstrip Length
MLIN1	70.71 Ω	90°	1.305 mm	54.43 mm

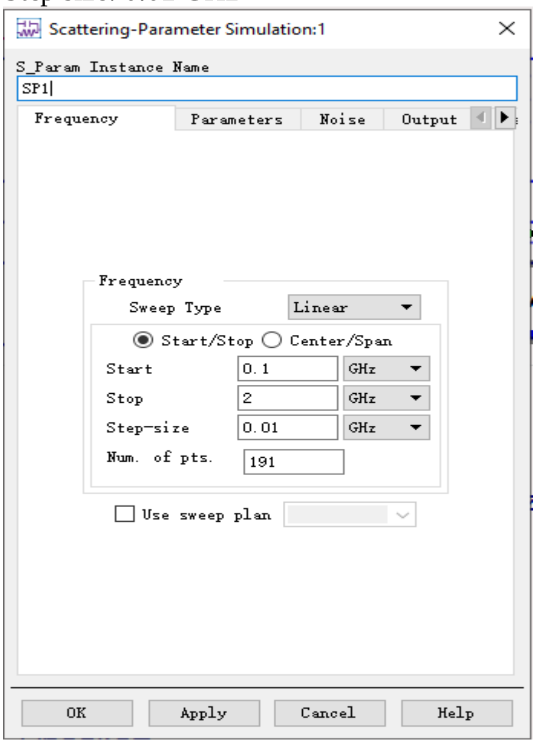
Setup and Run the Simulation

1. Select the **Simulation-S_Param** palette to insert the **S-parameter simulation** controller (gear icon). Then insert the S-parameter simulation controller on the schematic.

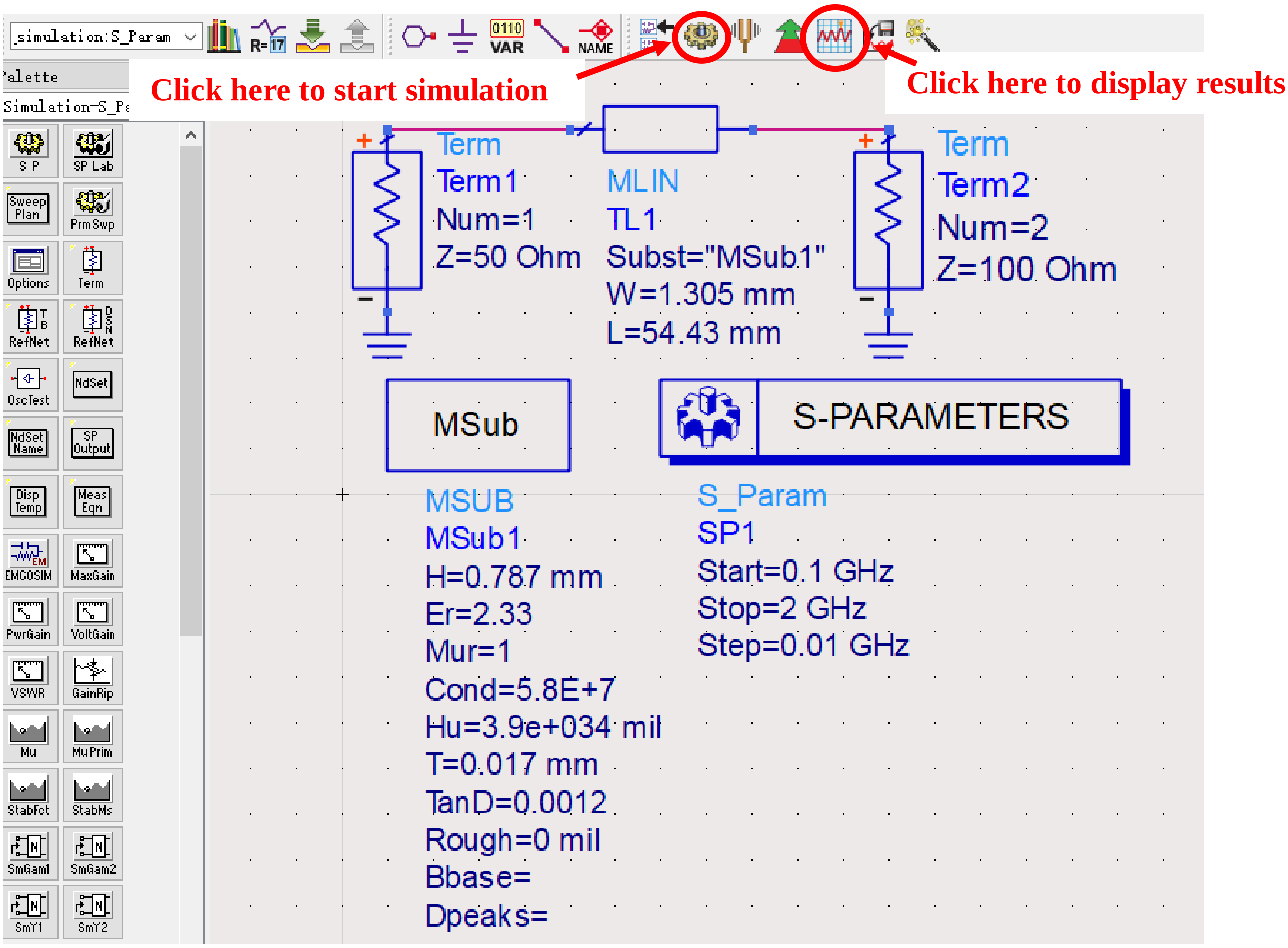


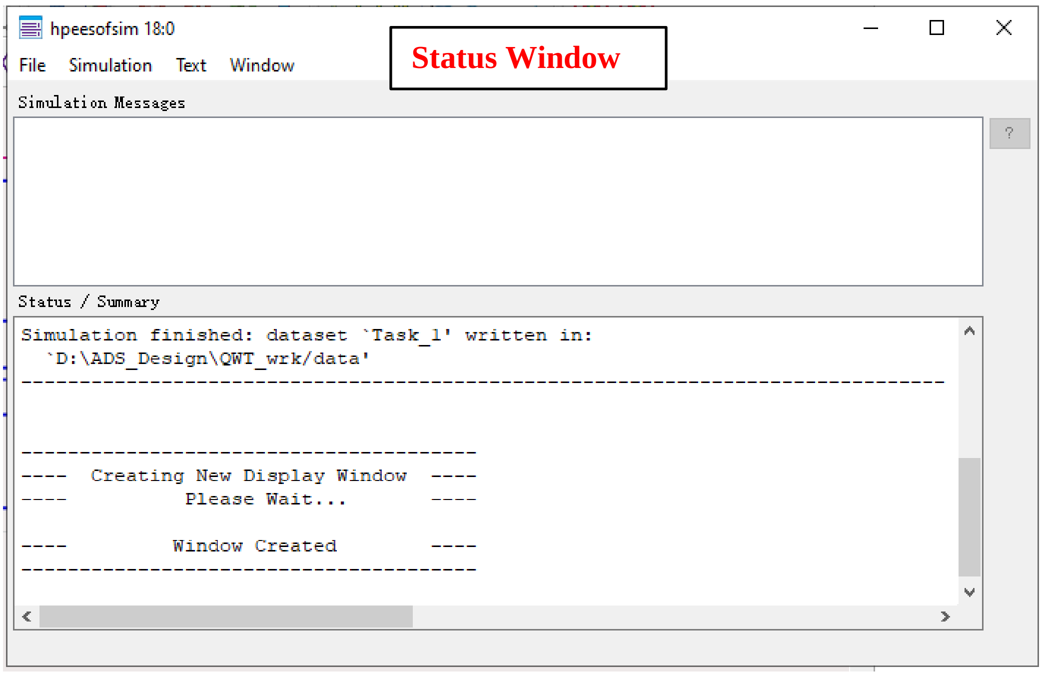
2. To setup the simulation, double-click on the **S-parameter simulation controller** on the schematic. When the dialog box appears, put the **start frequency**, the **stop frequency**, and the step size and click **Apply button**.

   - Start frequency: **0.1** GHz
   - Stop frequency: **2** GHz
   - Step size: **0.01** GHz

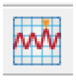


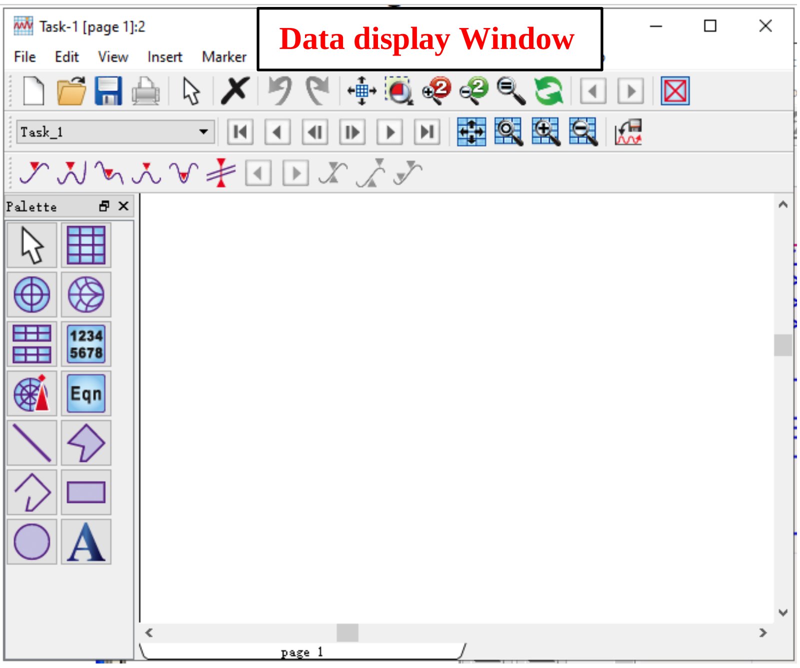
3. Click the **Simulate icon**  to start the simulation process.
4. Next, look for the **Status window** to appear and you should see a message that the **simulation finished**



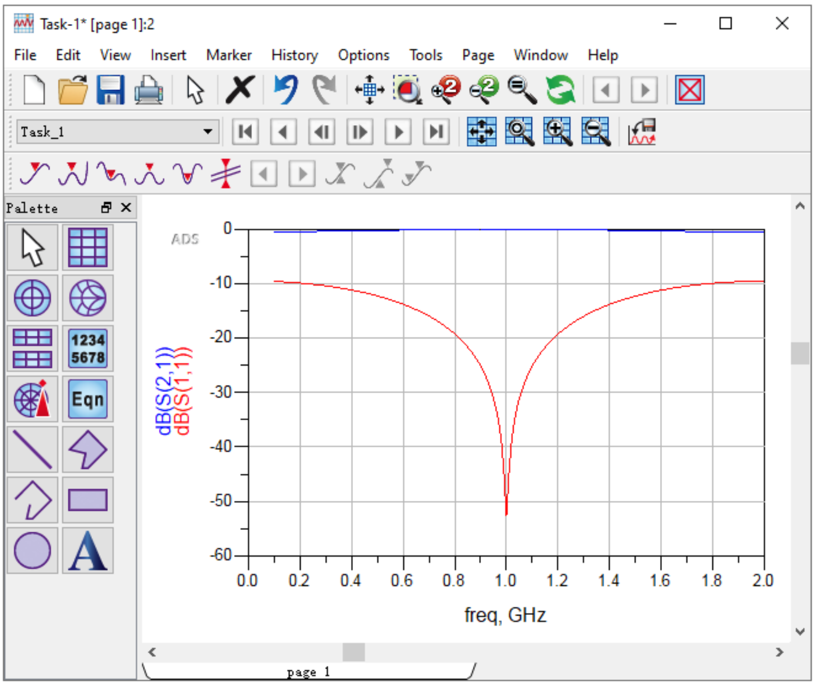


Display the simulation results (Data Display windows)

1. Open a data display window from either the Main window or the schematic window by clicking the Data Display icon .
2. When the Data Display window opens, the name of the dataset will appear in the list.



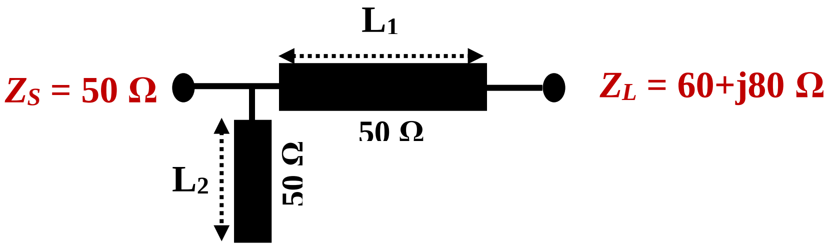
3. To create the plot, click on the **Rectangular Plot icon** and move the cursor (with ghost plot) into the window and **click**. When the next dialog box appears, select the S11 and S21 data and click the **Add button**.
4. The next dialog will prompt you to specify the type of data to display. Select **dB** and click **OK**. The plot should show an impedance transformer response.



Task-2

Design of L-Section Impedance Transformer

Design a L-section impedance transformer to transform a load impedance of $\boldsymbol{Z_L = 60+j80\ \Omega}$ to a source impedance of $\mathbf{Z_S = Z_0 = 50\ \Omega}$ at the center frequency of **2.0 GHz**. Microstrip technology can be used to develop the impedance transformer and Advanced Design System 15.01 software can be used to perform the simulation.



Procedure:

1. Calculate the electrical lengths L_1 and L_2 using Smith chart utility based on the load and source impedances.
2. Calculate the physical dimensions using **Lincalc** of ADS.
3. Design the schematic on the ADS workspace following the above.
4. Define substrate properties (consider same substrate as used earlier) and add frequency sweep as given.
5. Setup and run simulation.
6. Display the simulation data.

Smith Chart Utility

The Smith Chart Utility is accessed from the Tools menu or the Design Guide menu in the Schematic window.

To open the Smith Chart Utility:

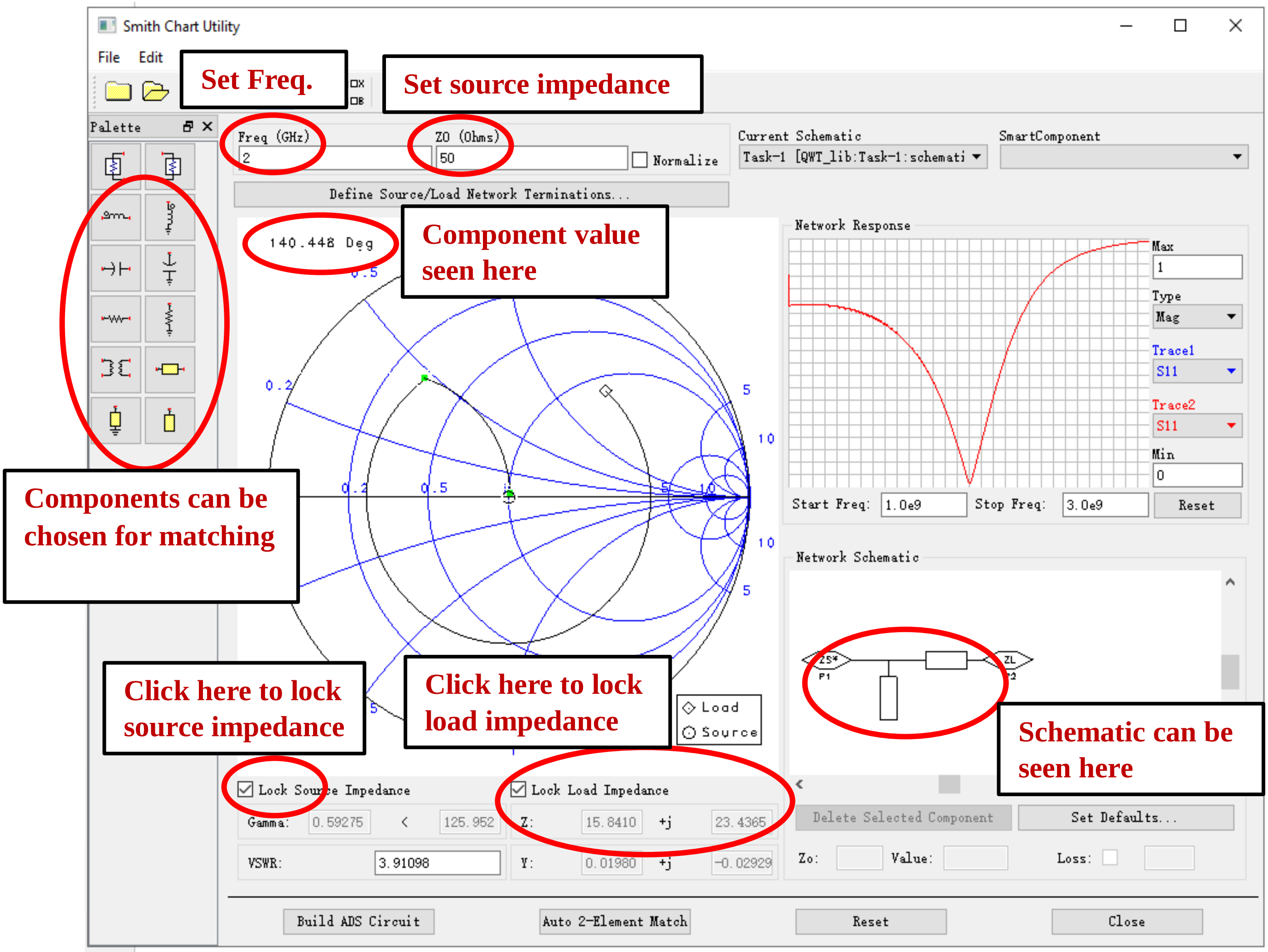
1. In the Schematic window, choose **Tools** > **Smith Chart**. The Smith Chart Utility window opens. Or, you can choose one of these paths from the Design Guide menu:

   DesignGuide > **Amplifier** > **Tools** > **Smith Chart Utility**

   DesignGuide > **Filter** > **Smith Chart**

   DesignGuide > **Mixers** > **Tools** > **Smith Chart Utility**

   DesignGuide > **Oscillator** > **Tools** > **Smith Chart**



1. Set the frequency as **2 GHz** as given.
2. Set the source impedance to **50 Ω** and load impedance to **60+j80 Ω**. Lock the source and load impedances. Now you can see the source and load impedances are locked on the smith chart.
3. Pick **MLIN** component and move on the smith chart, now you can see the value of the component on the top left corner based on the location you want to put the component.
4. Now we are off the center. To bring it to the center we have choose **MLOC** (open stub) and the corresponding value can be seen on the top left corner.
5. Based on these values calculate the physical dimensions using **Lincalc** and design a schematic on **ADS**.
6. Setup the frequency and run simulation.
7. Display the rectangular data in **mag** and in **dB**.

HA#3

Design impedance transformers to match a source of 50 Ω to a load of 100 Ω at a frequency of 2 GHz. Plot the reflection coefficient magnitude (also in dB) versus frequency for the designs using 2, 3, and 4 sections in a single rectangular plot.