Chapter 3

Program Menus, Options, and Settings

3.1 The Main Menu: Schematic Window

The File Menu:

- **New** clears the schematic page and prepares the program for a new schematic capture or synthesis session.
- **Open Schematic** pops up a "File Dialog Box" for opening an existing schematic.
- **Save Schematic** pops up a "File Dialog Box" for saving the currently displayed schematic.
- **View Netlist** shows the netlist for the currently displayed schematic.
- **Save Netlist…** saves the currently displayed schematic in netlist file format.
- **Open S-Param File…** loads S-parameters from a data file into the program for display or analysis. If the "Perform Single Frequency Analysis" box is left in the default (checked) state, then the Circles Utility will start with the specified S-parameter file. Unchecking this box loads the S-parameters into the program for direct viewing in the rectangular graph or Smith Chart windows. Select Options | Sweep to set up the band and number of (interpolated) frequency points over which to view the imported S-parameter data.
- **Save S-Param Data…** saves the S-parameter results of a circuit simulation to a data file.
- **Open *.CKT (Text) File…** Runs a simulation on a circuit file in netlist format. After opening a netlist (CKT) file, the simulation results can be displayed from the main View menu. The loaded netlist circuit file can be viewed by selecting Edit *.CKT (Text) File… from the main File menu.
- **Edit *.CKT (Text) File…** opens a full-screen text editor for generating and editing circuit netlists, data files, and other text files.
- **Print Setup…** accesses printer options.
- **Print Schematic** prints the currently displayed schematic window.
- **Exit Schematic** exits from the schematic page. Clicking **Exit** again quits the program entirely.

The Edit Menu:

- **Undo** reverses the last operation.
- **Copy Part** enables making a duplicate of any part already placed on the schematic page. After selecting **Edit | Copy Part,** press the left mouse button down over any existing part and drag its copy to a new position on the schematic.

Another way to duplicate a part is to first hold down the SHIFT key and then left click down over any part. Drag and drop the duplicate part to its new position.

- **Select All** selects all parts on the schematic. The outline of selected parts changes from black to red when selected. Parts selected can be deleted or moved as a group. Moving parts will delete their connections unless "Move All" is selected.
- **Deselect All** unselects all parts that were previously selected (parts outlined in red).
- **Move All** moves all parts on the page to a new position (drag-and-drop mouse action).
- **Move Selected** moves all selected (highlighted) parts to a new location on the page. The wire connections to all moved parts will be deleted.
- **Delete Last** deletes the last part placed.
- **Delete Selected** deletes all selected parts. Individual parts can be selected by clicking the right mouse button over the part. The color of a selected part changes to red. Clicking "Delete Selected" when no part is selected changes the shape of the mouse pointer to cross-hairs. Place the cross-hairs over the part and click either mouse button to delete the selected part. All connections to the part will also be deleted. The keyboard Delete key can also be used.

The Parts Menu:

The **Parts** menu displays the available parts and their placement orientations. Selecting a part from the menu drops it onto the schematic page and opens up a Part Parameter Dialog Box. The Part Parameter Dialog Box allows for editing the part's name, value(s), comments, and whether or not it can be tuned (or available for statistical analysis in the "Statistics" window). See Section 4.2 for part descriptions.

The Wire Menu:

Clicking on **Wire** starts placement of a wire connection between the terminals of a part or parts. Wire placement must begin by clicking the left mouse button over the terminal point of a part (little circle at the end of its lead). As the wire is routed, it can be anchored at any point on the schematic page (in order to change direction) by clicking the left mouse button. Multiple part pins (part terminals) can be connected at one time by continuing to click the left mouse button over each pin to be connected. However, the last pin to be connected at the end of the wire must be clicked with the *right* mouse button. To delete a wire, left click on it to highlight the wire in red and then press the Delete key. (In some cases all wires connected to the node may be deleted.) Note: It is not necessary to select the Wire menu to place a wire. The process of placing a wire connection can be started by simply left clicking on a part pin. **Wire placement starts by clicking the left mouse button on a pin and ends by clicking the right mouse button on a pin**.

The Set Menu:

The **Set** menu provides for the selection of various options and for controlling (or formatting) the output of the simulator.

- **Frequency Sweep** sets the frequency range and number of points over which the simulation will run (produce output).
- **Sweep Markers** sets the frequency markers that are used to display output data at four specific frequency points in the "GRAPH" window after the simulator has analyzed the circuit. If "Auto" is unchecked, then Markers 1 through 4 will be available for setting each frequency point manually. If "Auto" is checked, then all four markers will be set automatically at evenly placed points throughout the frequency range.
- **Grid** places a grid on the screen to aid in the placement and alignment of parts and wire traces. The grid can be turned on or off, or made light or dark.
- **Traces** provides control over the way wire traces can be routed on the schematic page. "Horizontal/Vertical Only" is the default and preferred method since it helps to keep the lines straight. However, occasionally it may be necessary or desirable to place a wire at a 45° angle or some other orientation. "Any Orientation" provides this flexibility if needed.
- **Output** pops up a "View Options" dialog box. This allows selecting what will be plotted in the "GRAPH" window after the circuit is analyzed. It also sets up the vertical scale(s) of the plotted data.
- **Snap to Pin...** adjusts the snap distance for connecting a wire to a part pin.
- **Show Nodes** displays the node number of each connection next to the part pins on the schematic page.

The Design Menu:

- **LNA (Circles Utility)** launches the Single Frequency Analysis (Circles) tool for analyzing and synthesizing low noise and maximum gain amplifiers.
- Clicking **Amplifier(s)** will synthesize a single or multistage amplifier schematic depending on the information entered into the Design Specifications form. If the schematic page is not initially blank, then File | New must first be selected to clear it.
- **Match** generates an impedance matching network between any unbalanced source and load port placed on the schematic page. The type of matching network is specified by the Input Matching tab of the Design Specifications form. If the schematic page is initially blank, then impedances to be matched will be found on the Ports tab of the Design Specifications form.
- **Convert T-Lines to…** converts certain transmission lines present in the schematic from their electrical representation to physical representations as microstrip or stripline.

- **Preferences/Design Specs** pops up the **Design Specifications** form for user control over the design characteristics of the amplifier or matching networks synthesized.

The Analyze Menu:

This menu button starts the simulation engine analyzing the circuit schematic. A successful simulation produces analysis results available for display by selecting the method from the View menu.

The View Menu:

- **Smith Chart** displays a graphical view of the complex (vector) quantities S_{11} and S_{22} for the analyzed circuit. The S-parameters from a data file can also be imported directly and displayed on the Smith Chart. To display S-parameters from a data file, select File | Open S-Param File…, uncheck Perform Single Frequency Analysis, and click OK. Marker data can be displayed at the bottom of the window in decibel or linear formats or as complex (real and imaginary) input and output port impedances. The display options can be changed by selecting View | Set Options or Options | Display.
- **Plot (Rectangular)** produces a rectangular plot of selected circuit responses or S-parameter data.
- **Results (Table)** presents a table of all circuit responses or S-parameter data.
- **Stability** displays stability circles for the circuit or S-parameter data at selected frequency points.
- **Statistics** opens a window for performing Monte Carlo statistical and yield analysis.
- **Set Options** opens a form for selecting the data to be displayed and the vertical scales for rectangular plots.

The Options Menu:

- **Set Colors** sets the trace colors of plots and grid lines.
- **Display** options are plot trace thickness, display grid lines on rectangular graphs, and display magnitudes in linear or decibel format.
- **Performance** sets the maximum number of simulation sweep points. The default number of points is 1001. However, setting this number lower, to 401 or 201 points, may enable the program to run on computers with less memory and processor speed.
- **Sweep** is used to set up the band and number of (interpolated) frequency points over which to view imported S-parameter data. Enter the Start Frequency, Stop Frequency, and number of points *after* importing an S-parameter file for analysis. Sweep does not set the simulation sweep frequencies. Use Set | Frequency Sweep… to specify the simulation sweep frequencies.

The About Menu:

- **About** displays the current version number and copyright information.

The Current File Menu:

When a previously saved schematic file is loaded from the File menu, the file name appears in the main menu bar just to the right of the About menu. Clicking on the file name in the menu bar pops up an Open File dialog box for loading another schematic file. When the schematic page is cleared, this **filename** menu is not displayed.

The Warnings Menu:

If an error occurs during simulation, a **Warnings** menu may appear as the last item on the main menu bar. The options listed are:

- **View** the simulation warning messages in a separate window with the following File options:
    - **Save Error File As…** saves the warnings to a disk text file.
    - **Print Error File** sends the warning messages to the printer.
    - **Exit Warnings** closes the message window and returns to the schematic page.
- **Clear** removes Warnings from the menu bar and clears the message file.

3.2 The Text Editor

A full-screen text editor is provided for generating and editing circuit netlists, data files, and other text files. When the Text Editor contains a circuit netlist, a circuit simulation can be launched directly from this window.

Selecting **Edit *.CKT (Text) File…** from the main File menu opens the Text Editor with the following menu options:

The File Menu:

- **New | Clear** clears the contents of the Text Editor.
- **New | Circuit Template** provides an outline of a circuit netlist. A text-based description of a circuit can be constructed by editing this file.
- **Open…** pops up an Open File dialog box for loading circuit (CKT) files into the editor.
- **Save As…** saves the contents of the Text Editor to a disk file.
- **Print Text** prints the contents of the Text Editor.
- **Analyze Circuit** launches a simulation of any valid circuit file displayed in the Text Editor's window. Upon successful simulation, a pop-up menu will offer various methods for viewing the results.
- **Circles Utility** launches a simulation of any valid circuit file displayed in the Text Editor and subsequently invokes the Circles Utility using the resulting S-parameters.

- **Exit** returns control to the schematic window.

The Edit Menu:

- **Cut** removes selected text and places it on the Windows clipboard.
- **Copy** places a copy of selected text into the clipboard.
- **Paste** copies any text from the clipboard to the current cursor location in the Text Editor.
- **Select All** selects and highlights the entire contents of the Text Editor.

The Analyze Menu:

This menu duplicates the function of **Analyze Circuit** in the File menu by launching a simulation of any valid circuit file in the Text Editor.

The Settings Menu:

- **Settings | Colors** sets the foreground (ForeColor) and background (BackColor) screen colors.
- **Font Sizes** sets the font size for displayed text.

The About Menu:

About… displays a message describing the intended purpose of the Text Editor.

3.3 The Single Frequency Analysis (Circles) Window

The File Menu:

- **Open S-Param File…** loads an S-parameter data file into the circles utility for analysis and design.
- **Save S-Param Data…** saves S-parameter data from the circles utility, including data modified in the utility (stabilized device data).
- **Print Window** prints an image of the window.
- **Exit** closes the Single Frequency Analysis (Circles) window without generating a schematic design.

The Frequency Menu:

- **Next** advances to the next analysis frequency.
- **Previous** steps back to the previous analysis frequency.
- **Select** displays a list of frequencies. Any frequency in the list can be selected for the current analysis frequency. If a desired frequency is not listed, then new frequency data can be entered via the Frequency | Sweep menu that captures the desired frequency point.
- **Sweep** defines a set of up to 21 frequency points. The start, stop, and number of frequency points can be specified. If any of the frequency points do not line up

with the frequencies in the original data file, then the program will interpolate (or extrapolate) the data at the frequency point that doesn't match the file.

The View Menu:

- **Stability** displays stability circles and related stability data for the selected frequency.
- **Gain, Ga** displays a constant available power gain circle on the input plane for a given gain and frequency selection. All points on the circle represent source reflection coefficients (Γ_S) that will produce the desired gain. These are mapped to a circle in the output plane. Each point on the output circle represents a conjugate output match (Γ_L) for every Γ_S (on the Ga circle) applied to the device input. Any point on the constant available power gain (Ga) circle can be selected by dragging the point around the circle with the left mouse button or by selecting CW Clockwise or CCW from the Move menu. The corresponding point on the output circle will be automatically updated. For every Γ_S and Γ_L combination selected, the gain remains constant. To select a different gain (and hence a different Ga circle), click View | Gain, Ga and enter a new value or right click anywhere within the input Smith Chart.
- **Gain, Gp** displays a constant operating power gain circle on the output plane for a given gain and frequency selection. All points on the circle represent load reflection coefficients (Γ_L) that will produce the desired gain. These are mapped to a circle in the input plane. Each point on the input circle represents a conjugate input match (Γ_S) for every Γ_L (on the Gp circle) applied to the device output. Any point on the constant operating power gain (Gp) circle can be selected by dragging the point around the circle with the left mouse button or by selecting CW Clockwise or CCW from the Move menu. The corresponding point on the input circle will be automatically updated. For every Γ_L and Γ_S combination selected, the gain remains constant. To select a different gain (and hence a different Gp circle), click View | Gain, Gp and enter a new value or right click anywhere within the output Smith Chart.
- **Noise and Ga:**
  - **Tracking** keeps the noise and gain circles locked and intersecting at a common impedance point. The impedance point (Γ_S, or source gamma in the input plane) can be selected anywhere within the Smith Chart by a simple click of the right mouse button. Both circles will snap to that point and the usual printed data in the information block at the bottom of the page will be updated.
  - **Independent** allows for controlling the gain and noise circles separately. The size of gain circles can be changed by right clicking on the Smith Chart (or dragging the gain circle to a larger or smaller value using the right mouse button). In addition, the mouse can control the impedance point on the gain circle. Just click any point on the circle with the left mouse button or drag the

impedance point around the circle. Noise circles can be increased or decreased in size by tapping the "+" or " -" keys.

- **Tradeoffs** lets you make gain or noise tradeoffs instantly. A slider control (or Scroll Bar) appears on the screen under the input Smith Chart. The left end of the scroll bar is designated "Max Gain" and the right side "Min Noise." Simply slide the bar to the left to increase gain or to the right to decrease noise figure. The gain and noise circles stay locked together as the source reflection/impedance point, Γ_S, moves along an optimum path between minimum noise and maximum gain.

- **Maximum Gain:**
  - **MAG (Gmax)** displays the maximum available gain (MAG) for the device. The MAG is unavailable if the device is potentially unstable. In most cases it can be stabilized unconditionally at the frequency of interest by selecting one of the methods from the Options | Stabilize Device menu. A point is plotted on the Smith Chart in each of the input and output planes. The points represent the Γ_S and Γ_L device terminations that will produce maximum available gain. The numerical values of the corresponding source and load reflection coefficients/impedances are printed at the bottom of the screen.
  - **Noise and MAG** produces all the data and displays as in MAG (Gmax) above with the addition of noise circle and noise data. The noise figure in decibels (when the device is terminated for MAG) is reported and the appropriate noise circle is plotted. The noise circle intersects the point $\Gamma_S = \Gamma_{max}$.
- **Minimum Noise:**
  - **Minimum Noise (Fmin)** plots the input match point ($\Gamma_S = \Gamma_{opt}$) for the minimum noise figure. The conjugate match point is also plotted in the output plane. The gain, noise, and matching impedances are printed at the bottom of the screen. Since the device is matched for minimum noise rather than maximum gain, the input VSWR will be greater than 1.
  - **Fmin and Gain (Ga)** produces all the data and displays as in Minimum Noise (Fmin) above with the addition of plotting a gain circle and related circle data.
- **Stability On** turns stability circles on (checked state) or off (unchecked state).
- **Freq Sweep** when checked enables plots of stability circles, gain circles, minimum noise match points and gain match points over a sweep of frequencies.
- **Clear** erases all plots and data displays and disables Freq Sweep.

The Move Menu:

- **Fast** sets the maximum step size for controlling gain and noise circles. Also sets the step size for traversing (moving the impedance match point) around circles.
- **Medium** sets the step size to medium for controlling gain and noise circles. Also sets the step size for traversing (moving the impedance match point) around circles.

- **Slow** sets the smallest step size for controlling gain and noise circles. Also sets the step size for traversing (moving the impedance match point) around circles.
- **CW (Clockwise)** moves an impedance match point around a gain circle in the clockwise direction.
- **CCW** moves an impedance match point around a gain circle in the counter-clockwise direction.

The Match Menu:

The Match menu produces the final output from the Circles Utility and then exits the utility. The result is a complete circuit synthesized with matching networks that extract the desired power gain and noise performance from the device. When source and load reflection/impedance points are plotted in the input and output planes, respectively, then the following matching networks can be selected from the Match menu to match the device. Upon selection of a matching network topology, a circuit netlist will be placed in the Text Editor or a schematic diagram of the circuit will appear in the schematic window (depending on the export method specified in the Match | Export Output menu).

- **L Match** provides all the available "L" match configurations for the input and output networks. Click "Select Input Match" or "Select Output Match" to specify the type of input, or output match, respectively.
- **T Match** specifies a T-matching topology at the input and output of the device. The phase shift through each network can be specified. This provides control over the component values and circuit Q.
- **PI Match** specifies a Đ-matching topology at the input and output of the device. The phase shift through each network can be specified. This provides control over the component values and circuit Q.
- **T-PI Match** specifies a T match at the input and a Đ-matching topology at the output of the device. The phase shift through each network can be specified. This provides control over the component values and circuit Q.
- **PI-T Match** specifies a Đ match at the input and a T-matching topology at the output of the device. The phase shift through each network can be specified. This provides control over the component values and circuit Q.
- **Transmission Line:**
  - **Single Section** matches the device input and output with a single section of transmission line.
  - **1/8 and 1/4 Wave Cascade** matches the device input and output with eighth-wave and quarter-wave transmission line sections.
  - **Stub and TRL Cascade** matches the device input and output with open or short stubs and transmission line elements.
- **Export Output** directs the circuit synthesis output to the following locations:
  - **To Netlist Editor**: The Text Editor receives the synthesis output.
  - **To Schematic:** The schematic window receives the synthesis output.

The Options Menu:

- **Target Z0in** sets the input port impedance. The input matching network transforms this impedance to the appropriate value required at the input terminal of the device. The default value for Z0in is 50Ù.
- **Target Z0out** sets the output port impedance. The output matching network transforms this impedance to the appropriate value required at the output terminal of the device. The default value for Z0out is 50Ù.
- **Stabilize Device**
  - **Series Load Resistor** uses a resistor in series with the output terminal to stabilize the device.
  - **Shunt Load Resistor** uses a resistor in shunt with the output terminal to stabilize the device.
  - **Common Lead Inductance** places an inductor between the common (otherwise grounded) device terminal and ground. This provides output to input series feedback.
  - **Reset S-parameters** removes the stability resistor or inductor and returns the S-parameter set to the original device or transistor's S-parameters.
- **Show G=1 Circle** displays the unity conductance circle.
- **Show Circles > MSG** removes the restriction of only allowing gain circles to be displayed up to the approximate maximum stable gain region on the Smith Chart. When a transistor is potentially unstable, it may be useful to explore gain circles and stability circles beyond the stable region. Of course, much of the matching conditions at or above MSG will be invalid for amplifier design.

The Current File Menu:

The current device file name appears as the last item on the menu. Clicking on the file name in the menu bar pops up an Open S-parameter dialog box for loading another data file.

3.4 The Monte Carlo Statistics Menus

The Yield Menu:

Calculates the **YIELD** based on the response goals and component tolerances. Up to four goals can be set up for each of two circuit responses. This allows for a circuit response to have different goals for different frequencies. The responses available are those previously set up from the main View | Set Options menu or through the Set | Output… menu prior to analyzing the circuit.

If Graph | Show Graph is checked, then the cumulative responses of all the Monte Carlo runs will be displayed in a graph window. Clicking YIELD from the graph window will return to the goal/yield window.

The Tolerances Menu:

Lists all the components that have been selected for tuning. During a Monte Carlo run, each element listed here will be varied uniformly over a range of values. Selecting any one from the list will pop up a Set Tolerance input box. A new tolerance (+/- percent value spread) can be entered, or the default +/- 5% uniform distribution can be accepted.

The Set Menu:

- **Sample Size** specifies the number of times the circuit will be evaluated while varying component values over their specified range.
- **Reset Tolerances** resets all circuit component variations to a specified range.

The Graph Menu:

- **Graph | Show Graph** displays cumulative responses of all the Monte Carlo runs in a graph window when the **Calculate Yield** button is pressed.
- **Graph | Return Statistics** exits the graph window and returns to the goal/yield window.

The Print Window Menu:

Prints an image of the current statistics or graph window.

The Exit Menu:

Exits the Monte Carlo Statistics window.

3.5 The Graph Menus

The Markers Menu:

- **Markers | Markers On** enables four markers to display response data at four frequency points. A Markers form displays the data numerically at each point.
- **Markers | Set Markers** sets the marker frequency for each marker.

The Tune Menu:

- The **Tune** list contains all the components available for tuning. The component's name and/or its nodal position in the circuit netlist is given. Selecting a component from the list displays its value in the graph window's menu bar. To change its value, click on the value displayed in the menu bar or tap the UP (↑) or DOWN (↓) arrow keys.
- **Reset Variables** returns all components to their value(s) before tuning.

The %Change Menu:

This menu changes the tuning percentage from the default value of 5%.

The Print Window Menu:

This menu prints an image of the graph window.

The Current Tune Component:

If a component has been selected for tuning from the Tune menu, then it will be displayed as the last item on the menu bar. Clicking on the current component in the menu bar pops up an input box for changing its value.

Clicking [X] closes the graph window. It is a good idea to turn Markers off by unchecking **Markers | Markers On** before closing the graph window. This helps to ensure that new marker data will be recalculated properly the next time a graph window is opened.