

# Quickstart Manual for PUFF

## Computer Aided Design for Microwave Integrated Circuits

EC415 – Wireless Electronics  
DRV—S2000

Copy the contents of the network copy f:\jupiter\almathea\ece\users\voltmer\public\ec415 or PUFF disk into the PUFF directory of your PC or laptop. Create a shortcut to Puff.exe; right click on the shortcut and select 'Properties'. Click on the 'Program' tab, enter the name of the required screen dump routine in the 'Batch file' window; for most printers use the routine VGA2LASR.COM. Check the 'Close on exit box at the bottom. Click on the 'Misc' tab; in the Windows shortcut keys section, make sure that 'PrtSc' is NOT checked. Click 'OK' box to save the settings and close the dialog window.

**Circuit File** – All PUFF analysis is based upon the parameters of a circuit file; this file can be saved for future use. To create *YOURFILE.PUF*, first copy the SETUP.PUF file (located in the PUFF folder) and paste it into *YOURFILE.PUF*. DO NOT ALTER THE ORIGINAL FILE. Second, edit *YOURFILE.PUF* to include the circuit board parameters, analysis frequencies, plot ranges, and circuit elements you will use. You can add circuit elements while running PUFF if needed. Third, save *YOURFILE.PUF* in a folder arrangement of your choice.

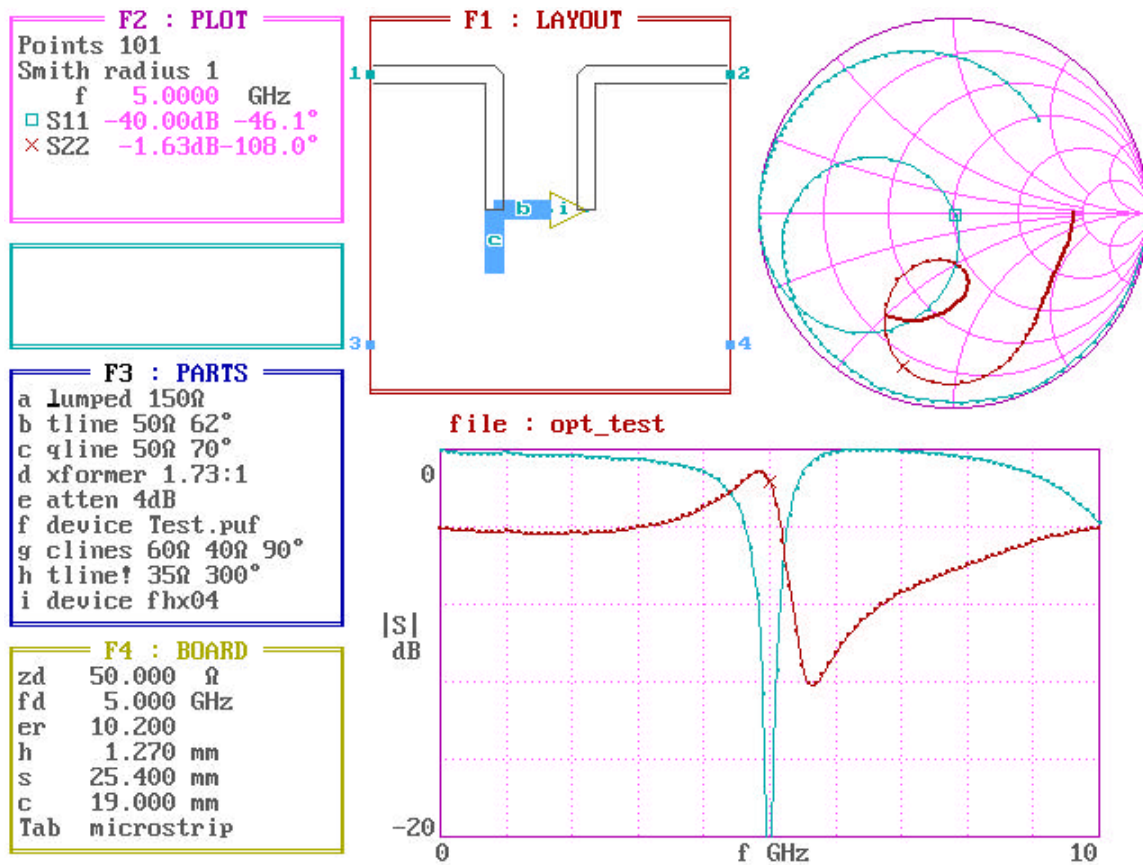


Figure 1 – Typical PUFF Screen

The PUFF screen (shown in Figure 1) is subdivided into 4 windows—LAYOUT, PLOT, PARTS, and BOARD—and a MESSAGE box. There are several hot keys that you use to navigate around the screen and to activate different functions.

#### HOT KEYS:

F1 –LAYOUT window: CURSOR keys are used within layout window to place highlighted part in PARTS window;

F2—PLOT window: Selected S-parameters are plotted on Smith Chart and rectangular dB plots;

F3—PARTS window: Parts for use in the layout window are specified;

F4—BOARD window: Board properties and type of line are displayed;

ESC—exit PUFF;

F10—Toggles HELP window with instructions (that appear in window 4 when windows 1-3 are active and in window 3 when window 4 is active) for the current active window;

CURSOR keys—Provide orientation of parts in the layout window and select, edit, and add entries within other windows;

and TAB key—Toggles window-specific features for windows 2, 3, and 4.

FUNCTION KEYS – Function keys 1 – 4 are always active and allow you to switch from window to window. The active window is indicated by the window number (at the top of the window) displayed in white letters.

**F1 – LAYOUT window** (LIGHT BLUE border, top center of screen): The location at which the next part will be added is a white x cursor; the default location for a new layout is the center of the board. The arrow keys are used for circuit layout with several different functions. Most of these functions are displayed within the HELP window (window 4) by using F10. A part is selected with letters ‘a’ through ‘r’ the LAYOUT window; the corresponding part is highlighted in the PARTS window. Each part in the LAYOUT window is identified by the same pink letter. (If the desired part is not in the PARTS window, then use F3 to switch to the PARTS window and add the desired parts.) The selected part is placed on the layout window in the direction of the arrow key used. The arrow can be moved back to the previous position with the opposite arrow-key. A SHIFT-arrow moves the cursor to a new position unless this motion takes it over an existing part in which case that part is erased. The amount of motion by the last cursor movement is displayed in the small (PINK) MESSAGE box located on the left side between the PLOT window and the PARTS window. Parts are shown in proper length and width according parameters in the circuit file. The circuit is connected to input/output ports by placing the cursor at the desired circuit node and entering the port number to which it is to be connected. The port numerals of connected ports are highlighted in pink. (Port connections can be erased with a SHIFTeD port number. When no port connections are present, a SHIFTeD port number takes the cursor to the port.) A CTRL-n brings the cursor to the closest node. A circuit node is grounded with the = key. The entire circuit can be erased with CTRL-e.

**F2—PLOT window** (GREEN border, upper left of screen): The S-parameter responses of the circuit in the LAYOUT window are activated within the PLOT window by the p-key. CTRL-p allows a second set of data to be plotted along with the first for comparison. A quick-plot mode

plots only after the analysis is completed; this mode is activated with the q-key and CTRL-q. The plot amplitude and frequency ranges are specified in the circuit file. The plotted data point at the center frequency,  $f_d$ , is denoted by the S-parameter symbol as shown in the PLOTS window and the numeric value is given in the PLOTS window. Changes in the PLOT window are facilitated with the arrow keys. The number of points can be specified within the PLOT window. The radius of the Smith chart can be changed from the default value of 1. The S-parameters to be plotted are entered into the plot window. The default Smith chart shows impedance coordinates; TAB toggles to admittance coordinates. After a plot (or quick plot) is completed, move the cursor to S11 or S22 and press the = key to display the series equivalent circuit (at  $f_d$ ) in the MESSAGE box when the Smith chart has impedance coordinates. TAB to admittance coordinates and repeat to display the parallel equivalent circuit. The vertical and horizontal scales of the plot are accessed by scrolling with the up/down arrow keys ( $\uparrow\downarrow$ ) and typing desired values. The cursor in the plot window can be moved with the PAGE UP and PAGE DOWN keys.

The impulse and step responses of a given S-parameter are calculated (by taking the FFT of the frequency response) and displayed in the rectangular plotting window with the i-key and s-key, respectively. The MESSAGE box prompts for specification of the ratio of the center frequency to frequency increment; the recommended default value of 10 is used with ENTER, but you may enter other values.

**F3—PARTS window** (YELLOW border, center left of screen): Parts for use in the LAYOUT window are specified in the PARTS window. Parameters can be altered or new parts entered with the aid of the arrow keys. Editing information is available with F10 help messages. The various elements and their specifications are shown in Table 1.

A few special notes on each type of part follow. Only the first letter of elements is needed.

**a(tten):** The attenuator is ideal, reciprocal, and symmetrical, has no frequency dependence, and is matched to the normalizing impedance  $z_d$ .

**x(former):** The ideal transformer is lossless and frequency independent. The turns ratio is always with respect to 1. Valid parameter specifications are 3.2:1 or 3.2. A negative turns ratio introduces a  $180^\circ$  phase shift. The wide end of the transformer (in the LAYOUT window) is the n side; the narrow end is the 1 side.

**l(umped):** The lumped part is used to specify series and parallel combinations of resistance, capacitance, and inductance. Four units of impedance are possible as designated by the suffix:  $\Omega$  (Ohms, ALT-o), S (Siemens), z (normalized impedance), and y (normalized admittance). Capacitance and inductance values are Farads (F) and Henries (H), respectively. Resistance, capacitance and inductance may be positive, zero, or negative in value. Reactance values are specified by placing a j before or after the a number and followed by  $\Omega$  or z. Numeric values of reactance represent the value at  $f_d$ . Positive reactances scale proportionally and negative reactances inversely with frequency. Series and parallel circuits are specified using + and || (ALT-p), respectively.

| <i>Part</i>    | <i>Description and Examples</i>   |
|----------------|---|
| <b>atten</b>   | Ideal attenuator. Enter attenuation in dB:<br>atten 3dB   |
| <b>xformer</b> | Ideal transformer. Enter turns ratio:<br>xformer 2:1  |
| <b>lumped</b>  | Resistor, capacitor, and/or inductor:<br>lumped 50Ω  1nH  1pF {parallel RLC}<br>lumped 10+j10-j10Ω {series RLC resonant at fd}  |
| <b>tline</b>   | Ideal transmission line. Enter impedance and length:<br>tline 50Ω 90° {λ/4 length at fd}<br>tline 1.0z 5mm +1.0h {artwork length correction added}<br>tline! 50Ω 90° {analyze with advanced models}           |
| <b>qline</b>   | Finite Q tline:<br>qline 50Ω 90° 75qd {Q = 75 due to dielectric losses}<br>qline 20mS 90° 50qc {Q = 50 due to conductor losses}   |
| <b>clines</b>  | Ideal coupled transmission lines. Enter $Z_e$ , $Z_o$ , and length:<br>clines 60Ω 90° { $Z_e$ specified}<br>clines 60Ω 40Ω 6mm { $Z_e$ and $Z_o$ specified}<br>clines! 60Ω 90° {analyze with advanced models} |
| <b>device</b>  | Read file with s-parameter data. Specify filename:<br>device fhx04 {read filename fhx04.dev}<br>device atf131.s2p {read file in EEsop format}   |
| <b>indef</b>   | Indefinite s-parameter generator. Converts n port s-parameter file to n + 1 port device. Specify filename as in device.   |

Table 1 – Description of parts available in PUFF.

**t(line)**: The tline is an ideal, lossless, dispersionless transmission line specified by characteristic impedance,  $z_d$ , and length in meters (m), substrate thickness (h), or degrees (ALT-o) at the design frequency,  $f_d$ . With the cursor on a tline part, pressing the = key displays the length and width of the tline in the MESSAGE box. If no artwork is needed and the circuit is for analysis only, the length of transmission lines can be forced to one tenth of the board dimension in MANHATTEN mode via TAB in the BOARD window. Artwork corrections to compensate for discontinuities are allowed; see the full manual.

**q(line)**: The qline is similar to tline, but is lossy. The loss is specified through the quality factor of the line, Q, at  $f_d$ . At other frequencies the behavior is assumed to be due to dielectric loss or can be explicitly specified as Qd. Alternatively, conductor loss can be modeled as Qc.

**c(lines):** A pair of lossless, coupled transmission lines is specified much as tline except that it requires two impedances, even and odd modes, as well as a length. An alternate specification only uses one impedance. If it is larger than  $z_d$  it is interpreted as even-mode impedance,  $z_e$ , with the odd mode impedance,  $z_o$ , calculated as  $z_o = z_d^2 / z_e$ . If the single impedance is less than  $z_d$ , it is interpreted as odd mode impedance and the same equation is used to compute  $z_e$ .

**d(device):** The S-parameters for a device (manufacturers specs, measured data, or theoretical, idealized parts) are entered into a circuit via the device part followed by the file name containing the parameters, *DEVICE\_FILE\_NAME*; a .dev extension is assumed. An optional physical length may follow the file name. A previously executed circuit file that includes saved S-parameters can be used as a device file, but requires explicit inclusion of the .puf extension. Complicated circuits can be analyzed by combining many smaller circuits in this way. Follow the file fhx04.dev as a template for making your own files. See the full manual for details in making device files. Device files of the form used by EESOF are also recognized by PUFF.

**i(ndef):** The indefinite S-parameters for a device are calculated from the device file. The original device parameters can be reproduced by grounding the internal port shown as a YELLOW dot within the device. See the full manual for details.

Circuit files can be saved (CTRL-s within the PLOTS window) and retrieved (CTRL-r within the PARTS window). The default directory is PUFF, but any folder can be used with full path specification, e.g., c:\puff\circuits\ckt1. The extension .puf is added automatically.

In the PARTS window the = key displays the dimensions of the highlighted part in the MESSAGE window. Devices without wavelength dependence (atten, xformer, and lumped) are assumed to have a length of one-tenth the board size unless a dimension parameter is entered after the last electrical parameter for the device. Upon switching out of the PARTS window, error messages appear in the MESSAGE box for parts that are longer than the board at the specified center frequency,  $f_d$ , or shorter than the resolution,  $r$  (available only in the circuit file).

**F4—BOARD window** (DARK BLUE border, lower left of screen): The properties of the board material are specified here. In addition, the center frequency,  $f_d$ , and the normalizing impedance,  $z_d$ , are also entered here.  $f_d$  should be in the plot range specified within the circuit file. Port (or connector) spacing is controlled by the parameter  $c$  in the BOARD window.

Circuit Board Artwork—Typing CTRL-a from the PLOT window will activate PUFF's photographic artwork routines. The layout produced will be magnified by the photographic reduction ratio ( $p$ ) in the circuit file. The artwork output parameter ( $o$ ) in the circuit file allows you to specify dot-matrix or HP LaserJet printouts or the production of a HP-GL file. Puff will prompt for titles to be placed atop the printout or for an HP-GL file name.